Army Research Laboratory

RESTRAN: Residual Strength Analysis of Impact Damaged Composite Laminates Volume II: User's Manual

by Erik Saether

ARL-TR-2550 July 2001

Army Research Laboratory
Aberdeen Proving Ground, MD 21005-5069

ARL-TR-2550 July 2001

RESTRAN: Residual Strength Analysis of Impact Damaged Composite Laminates Volume II: User's Manual

Erik Saether
Weapons and Materials Research Directorate, ARL

Abstract

A general predictive methodology for determining residual strength in impact damaged composite laminates has been developed and incorporated into a computer code designated RESTRAN (REsidual STRength ANalysis). RESTRAN is a finite element based design tool that can analyze composite structures with arbitrary three-dimensional (3-D) geometry, loading and support conditions, material properties, and initial material and delamination damage. Material failure modes are predicted using a robust suite of failure criteria and damage laws. Structural failure due to sequential sublaminate buckling of delaminated layers is also accounted for. A progressive failure analysis is performed until ultimate structural failure is predicted, thereby yielding an estimate of the residual strength. This report contains a user's manual for the RESTRAN program with complete descriptions of input statements and program output. Several examples are shown to illustrate the use of the RESTRAN computer code.

Table of Contents

INTENTIONALLY LEFT BLANK.

List of Figures

List of Tables

INTENTIONALLY LEFT BLANK.

1 Introduction

The development of the RESTRAN (*RE*sidual *STR*ength *AN*alysis) computer program represents a significant contribution as a design tool to predict the reduction in structural strength due to impact damage. The numerical finite element basis provides the greatest versatility for modeling structures of arbitrary geometry, support, and load conditions while accounting for both material and structural failure modes. Although designed specifically for the analysis of advanced fibrous composites, the support of a layered media permits additional material systems to be modeled such as sandwich-type constructions, piecewise linear approximations of homogeneous materials with varying properties along a thickness direction, and isotropic materials modeled as a single layer. For composites, material failure may be differentiated into fiber and matrix modes, while structural failure involves buckling of local sublaminates due to the presence of delamination damage. In addition to built-in features, a suite of user-defined subroutine interfaces are available for problem-specific tailoring of the analysis. Optional graphics may be requested and output as input files for display using MATHEMATICA [1], TECPLOT [2], or in a user-defined format. The analysis capabilities of RESTRAN are thoroughly discussed in the Theoretical Manual [3]. The current report details the use of the RESTRAN code. Complete descriptions are given of the program input statements to define the structural geometry, support and load conditions, material behavior, and solution procedures. Finally, a series of demonstration problems showing program input and output are provided to illustrate the use and capabilities of RESTRAN.

2 RESTRAN Input Statements

The following sections detail the syntax and basic format of input statements available to describe the model geometry and invoke program options. Some degree of flexibility has been implemented in the parsing of these statements such that either upper or lower case letters can be used, and arbitrary space or comma delimitators can be used in listing statement options and data items. The basic format of all input statements is:

$$\textbf{*STATEMENT, OPTION}_1 = \textbf{X, OPTION}_2 = \textbf{Y, ... , OPTION}_n = \textbf{Z}$$
$$\textbf{DATA}_1\textbf{, DATA}_2\textbf{, DATA}_3\textbf{, ... , DATA}_n$$

One requirement, however, is that the ***STATEMENT** entry must begin in the first field in the record or line of the input file. The input statements may generally occur in any order and are parsed sequentially as contained in the input file. Statements may be repeated to supersede prior instructions or to append additional quantities to the input database such as nodes or elements.

In developing syntax for RESTRAN input, many of the primary statements such as node, element, boundary, and mesh generation are based on the syntax used in the commercial finite element analysis programs ABAQUS [4]. Other statements have a parallel in NASTRAN [5]. This was implemented so that preprocessor programs such as PATRAN [6], which facilitate the preparation of input for these commercial codes, could be used to create the majority of the input data for RESTRAN. The addition of a small set of RESTRAN-specific statements completes the input description to perform the various analysis options. The input statements supported are listed in Table 1.

Table 1: RESTRAN input statements.

STATEMENT	CATEGORY	PAGE
* PREPASS	EXECUTIVE	3
* SOLUTION		3
* MEMORY ALLOCATION	SYSTEM RESOURCES	5
* NODE	MODEL DESCRIPTION	5
* NGEN		6
* NFILL		6
* ELEMENT		7
* ELGEN		8
* NSET		8
* NSET2		8
* ELSET		9
* ELSET2		9
* BOUNDARY		9
* BOUNDARY2		9
* CLOAD		10
* PRESSURE		10
* MODEL GENERATION		11
* LAYER		17
* MATERIAL		17
* NONLINEAR MATERIAL		18
* FAILURE CRITERION		21
* DAMAGE LAW		24
* INITIAL DAMAGE		25
* DELAMINATION		27
* POSTBUCKLED MATERIAL		27
* ORIENTATION		28
* DEFORMED GEOMETRY		29
* EQUIVALENCE		29
* INCLUDE		37
* EXCLUDE ELEMENT		30
* ENDDATA		30
* PARAMETER USRBKL	EXECUTION MODIFICATION	31
* PARAMETER STATUS		33
* PARAMETER LAPFAIL		33
* PARAMETER DIRECTORY		34
* NODE PRINT	OUTPUT CONTROL	34
* ELEMENT PRINT		34
* PLY FAILURE PRINT		34
* GRAPHICS		35
* ECHO		37
* HEADING		37

2.1 Executive Control

Two statements are available to dictate executive control over the execution of RESTRAN. These statements are used to invoke preliminary model checking or initiate various analysis procedures.

2.1.1 Preliminary Model Evaluation

To perform a detailed check on the inputted model, the following executive control statement can be included anywhere in the input stream.

***PREPASS**

This statement causes RESTRAN to process the input file, perform error checking, assess memory requirements, and generate optional graphics of the model. This option overrides any other specified solution procedure and is used to assess the validity of the inputted model prior to performing any further analysis.

2.1.2 Analysis Procedures

Several solution procedures are available in RESTRAN. The executive control statement to request specific analysis procedures is given by

***SOLUTION, METH = PROC**
$\mathbf{LIM_1}$, $\mathbf{LIM_2}$, ACCEL
PROC, ITERB, CVAL, MTOL, RTOL

This statement selects the analysis procedure to be invoked by RESTRAN by assigning the mandatory parameter **METH**. Two linear analyses and three residual strength procedures may be selected. For linear static analysis, the method parameter is set as

METH = LSA

This directs RESTRAN to perform a linear static analysis to determine the basic elastic response of the structure. This option may be used to check displacements, stresses and strains, and an optional graphical depiction of deformation state, to assess applied loads and support conditions imposed on the model. In addition, the inclusion of the ***PLY FAILURE PRINT** option will output first-ply failure statistics. The data list following the ***SOLUTION** statement can be omitted.

METH = LBA

This option directs RESTRAN to perform a linear buckling analysis. This procedure may be used to analyze stability in a multiply-delaminated composite with all delaminations considered simultaneously. The predominant buckling mode can then be determined such that only a single, critical delamination may be input for a subsequent residual strength analysis. This would greatly speed execution because the residual strength algorithm will consider each delamination in sequence to automatically choose the critical delamination involved in sublaminate buckling. Optional graphical output can be selected to view the buckled mode shape. As with the **LSA**

solution procedure, the data list following the ***SOLUTION** statement can be omitted.

To invoke an incremental/iterative residual strength analysis, the following options are available.

$$METH = \begin{cases} MTL \\ BKL \\ CMB \end{cases} \tag{1}$$

These options specify the range of failure modes to be accounted in the analysis. The **METH** = **MTL** parameter is used to specify that only material failure modes are to be processed. **METH** = **BKL** causes only sublaminate buckling failures to be analyzed, and **METH** = **CMB** directs the solution algorithm to consider both material and buckling failure modes.

The control parameters set basic iteration bounds in the residual strength analysis. $\mathbf{LIM_1}$ is the maximum number of sequential load increments to be calculated in the prediction of residual strength. These loads are automatically determined as scalar factors applied to the initial set of external loads, and this limit may be set arbitrarily high. In certain cases where user-defined subroutines are used to degrade material properties undergoing failure, arbitrary degradation laws may be input which could, in some situations, cause internal tests performed by RESTRAN to avoid determining catastrophic failure. An upper bound on the number of total analysis cycles is therefore needed to stop runaway execution. $\mathbf{LIM_2}$ is thus used to set the maximum number of iterations to converge the damage state at the current load level. For each iteration, failures are accounted and internal loads redistributed, which may cause additional failures to occur. Although most failure events tend to converge rapidly to an equilibrium damage state or cascade to ultimate structural collapse, this limit sets bounds on possible slow failure cascades which may warrant changes in other input parameters. The optional **ACCEL** parameter is used to accelerate the prediction of failure by encompassing larger groups of local failures per analysis cycle. RESTRAN automatically determines a scale factor applied to the vector of initial input loads to identically satisfy selected failure criteria for a critical ply in a particular element. This set of loads is then increased by a default value of 1% to cause other plies to fail which may have calculated failure indices near unity. The value of the **ACCEL** parameter overrides the default value such that, for example, an input value of 1.1 will cause all plies within 10% of the critical ply load to fail in the current cycle and thus accelerate the prediction of local failures. Care must be given in setting the value of this parameter, as it directly modifies the applied loads and therefore the predicted value of the residual strength. A value less than unity would preclude material failure and is therefore automatically reset to the default value. A large value would cause an overprediction of the residual strength and would have to be manually factored out of the final predicted strength measure by the user.

For residual strength analysis involving sublaminate buckling failure, **PROC** may be set equal to 'SIMULTANEOUS' to analyze all delaminations as a single group or 'ITERATIVE' to analyze each delamination separately and to iterate ITERB times to obtain a solution to the constrained buckling contact problem of an embedded delamination. To approximately solve the contact constraints involved with sublaminate buckling, iterations are performed to minimize the degree of interpenetration of a buckling mode to surrounding layers. This is performed by repeated solution of a linear buckling analysis of a particular delamination. The complete satisfaction of constraints is impractical and unnecessary to obtain reasonable local buckling loads. The parameters **CVAL**, **MTOL**, and **RTOL** specify specific treatment of delamination buckling and range from 0 to 1 due

to an internal normalization of the buckling mode vector. To limit the number of iterations and to specify a criteria for accepting a particular buckling mode, the fraction of nodes exhibiting a physical opening mode is required. The default value of **CVAL** is 0.9, indicating that a mode is accepted if only 10 % of nodes in the delamination plane are involved in nonphysical interpenetrating displacements. The entry **MTOL** indicates the ratio of maximum local modal displacements across the delamination plane with the maximum overall buckling mode displacement. This ratio is used to predict whether a mixed-mode instability is occurring wherein the delamination being analyzed may be undergoing an opening mode; however, the overall structure is exhibiting a greater deflection indicating a global mode. The default value for **MTOL** is 0.5. Setting this tolerance to zero will preclude a mixed global-local buckling mode from being predicted. Finally, the **RTOL** entry indicates the threshold below which the relative motion of opposing delamination surfaces are to be considered zero, and coincident nodes are to be condensed out of the current delamination plane. Unless specified, the default value for **RTOL** is 0.0.

2.2 System Resources

A single large array is used within RESTRAN to perform numerical operations on global-sized matrices. This in-core array is used to provide working storage to perform fast matrix operations with the bulk of generated internal data stored in external scratch files. In the normal default execution mode, RESTRAN uses a half bandwidth storage scheme internally. In order to run large jobs, an out-of-core solution algorithm may be invoked. This causes additional processing and will slow down the execution speed. The internal algorithm for memory usage is specified using the following entry:

***MEMORY ALLOCATION**
STYPE

where **STYPE** = **'BANDWIDTH'** for half bandwidth storage and **'OUT-OF-CORE'** for fully external storage. A storage format of **'FULL'** is recognized which causes all operations to be performed using full matrix storage. The only practical use of this feature is to speed execution of small problems in which the connectivity unavoidably yields a global stiffness matrix that is densely populated as opposed to sparse. If this statement is omitted, the default storage mode is **BANDWIDTH**.

2.3 Model Description

The following subsections detail the RESTRAN input statements used to define a structural model.

2.3.1 Node Definition and Generation

Nodes are input using the following statement:

***NODE, SETID = NSD**
NODE ID, X, Y, Z

where the node number and position in global x, y, z coordinates are repeated for each node

in the model. The optional parameter **SETID** is used to include the specified nodes into a new or existing set. The node set ID must be numerical; general character strings are not supported.

To automatically generate nodes, two utility routines may be invoked to generate a linear and planar field of nodes. These are specified by the commands ***NGEN** and ***NFILL**.

To generate a linear array of nodes, the following statement is used

***NGEN, SETID = NGI**
$\mathbf{NODE_1}$, $\mathbf{NODE_2}$, INCR

As shown in Figure 1, ***NGEN** generates a line of nodes between two previously defined nodes, $\mathbf{NODE_1}$ and $\mathbf{NODE_2}$. The increment value, **INCR**, specifies the increment in the node numbers of equally spaced nodes to be generated between the two existing nodes. This increment thereby specifies the number of nodes generated, which is given by

$$NUMGEN = (|NODE_2 - NODE_1|/INCR) - 1 \quad (2)$$

The generated nodes may optionally be assigned to a node set by including the **SETID** parameter. This option must be specified if the ***NFILL** command is to be used with these generated nodes.

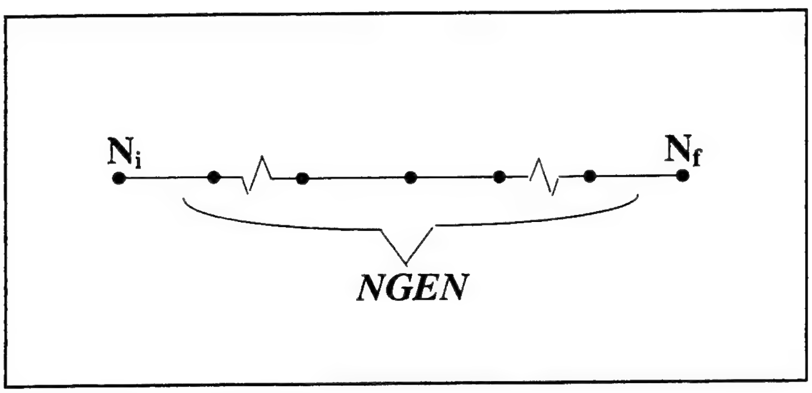


Figure 1. One-dimensional node generation.

A two-dimensional plane of nodes may be generated using the following statement:

***NFILL, SETID = NFD**
$\mathbf{NSET_1}$, $\mathbf{NSET_2}$, NUMGEN, INCR

As shown in Figure 2, two lines of nodes with specified set identification numbers $\mathbf{NSET_1}$, $\mathbf{NSET_2}$ are used to generate a plane of nodes. In generating the two-dimensional field of nodes, **NUMGEN** indicates the number of equally spaced nodes to be generated between bounding nodes in each set, and **INCR** indicates the increment in node numbering between each set. The node set parameter is optional.

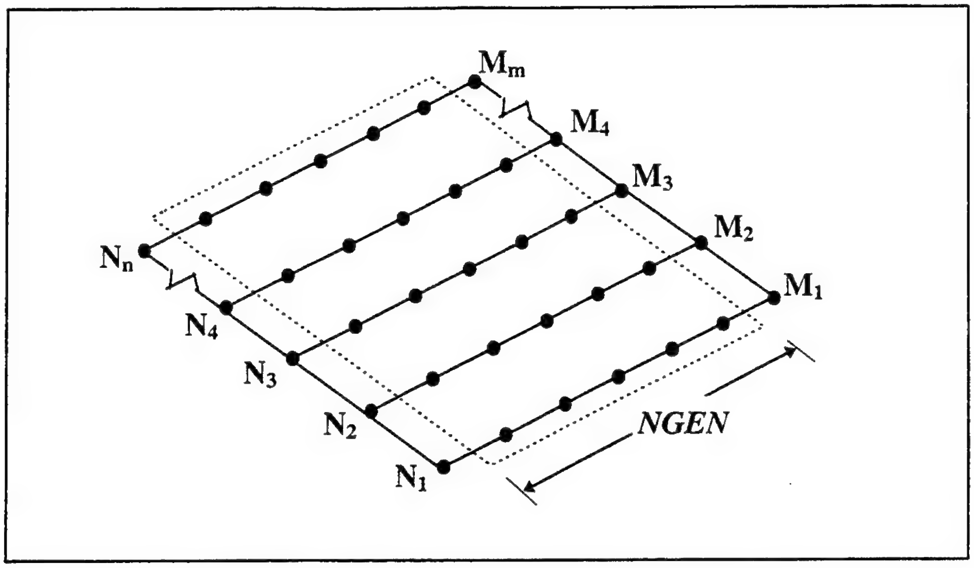


Figure 2. Two-dimensional node generation.

2.3.2 Element Definition and Generation

Element definitions are input using the following statement:

***ELEMENT, LAYUP = L, ORIENTATION = M, ELSET = EID**
EID, N_1, N_2, N_3, N_4, N_5, N_6, N_7, N_8

The data list gives the element number and the node numbers of the nodes defining the element geometry as shown in Figure 3. The mandatory parameter, **LAYUP = L**, associates the elements with a particular input sublaminate. The optional orientation parameter designates the identification number of an inputted local coordinate system to be assigned to these elements. The **ELSET** parameter is also optional to assign each inputted element to an element set. The data list is repeated for all elements associated with the specified layup and local coordinate system. The ***ELEMENT** data block is repeated for each group of elements with a unique assigned layup and spatial orientation.

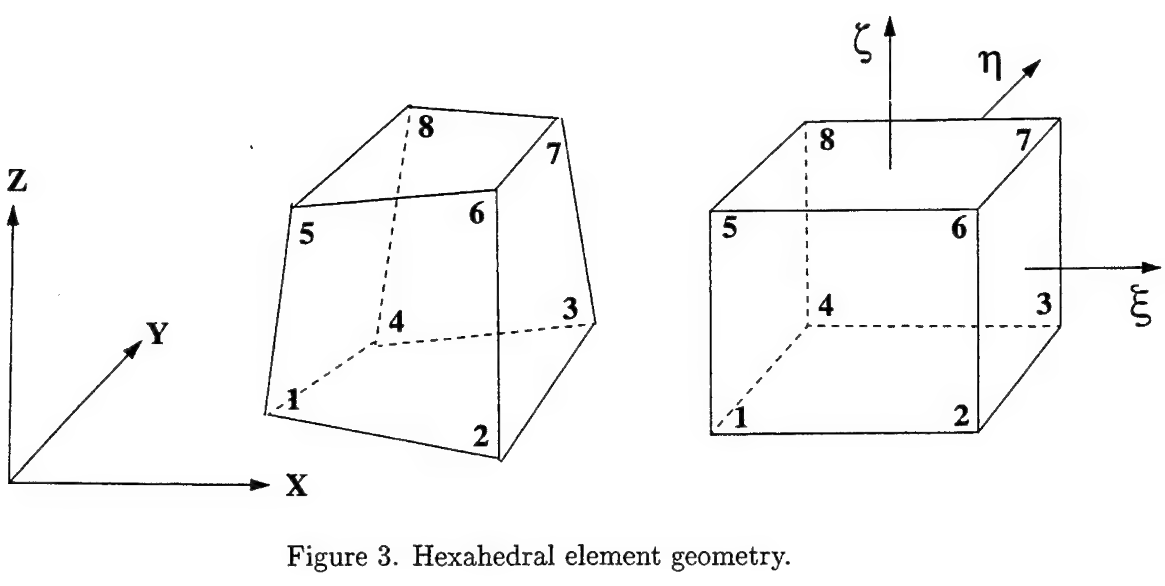


Figure 3. Hexahedral element geometry.

Automated generation of elements is supported by the following input statement:

***ELGEN, ELSET = EID**
ELID, $\mathbf{NGN}_X$, $\mathbf{NOD}_X$, $\mathbf{NID}_X$, $\mathbf{NGN}_Y$, $\mathbf{NOD}_Y$, $\mathbf{NID}_Y$,$\mathbf{NGN}_Z$, $\mathbf{NOD}_Z$, $\mathbf{NID}_Z$

ELID indicates the starting element from which other elements are to be generated. The parameters $\mathbf{NGN}_{X,Y,or Z}$ indicate the number of elements to be generated along the global X, Y, or Z directions, $\mathbf{NOD}_{X,Y,or Z}$ indicates the increment in node numbers, and $\mathbf{NID}_{X,Y,or Z}$ indicates the increment in element numbers along the X, Y, or Z directions, respectively. The optional parameter **ELSET** may be used to add the generated elements to a new or existing element set. Element generation is depicted in Figure 4.

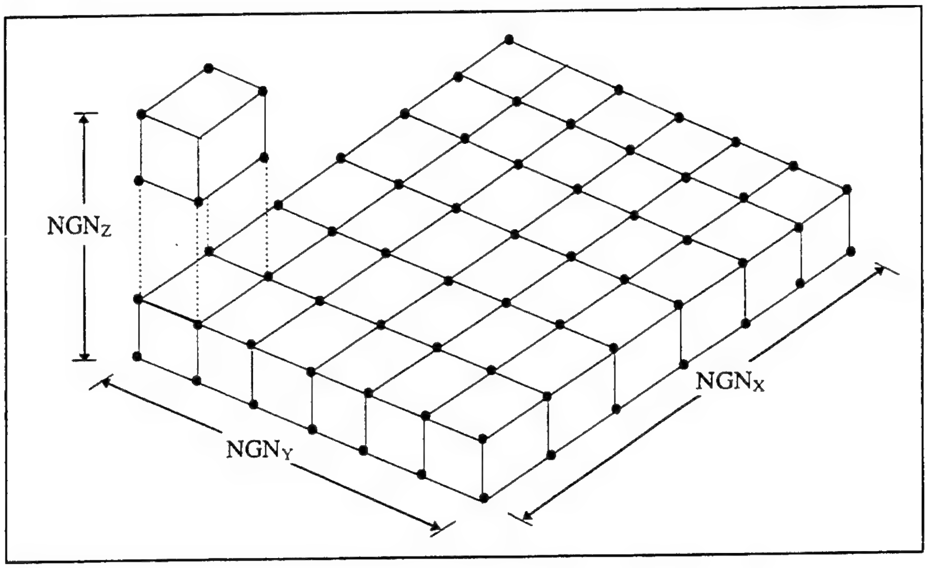


Figure 4. Generation of a one-, two- or three-dimensional field of elements.

2.3.3 Node and Element Sets

Node and element sets are used to define delamination planes, initial damage, and in print request statements. Node sets are declared using the following statements.

***NSET, SETID = NID**
$\mathbf{N}_1$, $\mathbf{N}_2$, ... ,$\mathbf{N}_k$

Where **SETID** = **NID** gives the set identification number, and $\mathbf{N}_1$, $\mathbf{N}_2$, ... ,$\mathbf{N}_k$ are the set of node numbers. The list of nodes can contain any number per line delimited by either commas or blanks and continued on an arbitrary number of additional lines. An alternate statement is available for nodes defined within bounds.

***NSET2**
NID, $\mathbf{N}_1$, $\mathbf{N}_2$

where **NID** is the set number, and $\mathbf{N_1}$ and $\mathbf{N_2}$ give an inclusive range of node numbers to be formed into a set. Only nodes within the given range that are actually used in the model will be stored. This line may be repeated to define multiple sets.

The ***NSET** and **NSET2** statements can be used to exclusively define node sets or used in conjunction with the optional node set parameters on the ***NODE**, ***NGEN**, or ***NFILL** statements. Depending on the order in which the input records are parsed, any statement can initiate the creation of a new node set while subsequent statements referencing the same node set will be accumulated into the existing set.

Element sets are defined using the following command.

***ELSET, SETID = EID**
$\mathbf{E_1, E_2, E_3, \ldots ,E_k}$

where **EID** is the assigned element set number, and $\mathbf{E_1} \rightarrow \mathbf{E_k}$ is the sequence of element numbers to be included in the set. The number of element ID's per line is arbitrary, and any number of additional lines may be used to complete the set definition.

Element sets may additionally be defined using the following command.

***ELSET2**
EID $\mathbf{E_1, E_2}$

where **EID** is the assigned element set number, and $\mathbf{E_1}$ and $\mathbf{E_2}$ give a range of element numbers to be included in the set. An arbitrary number of element sets may be defined using this statement.

As with node sets, element sets may be defined or built up using any combination of ***ELEMENT**, ***ELGEN**, ***ELSET**, and ***ELSET2** statements.

2.3.4 Boundary Constraints and Load Conditions

Specific point constraints on displacement degrees of freedom may be applied to enforce fixity conditions at any node. Two input statements are available to apply these conditions.

***BOUNDARY**
NODE, $\mathbf{DOF_1, DOF_2}$

where **NODE** is the node number. $\mathbf{DOF_1}$ and $\mathbf{DOF_2}$ specify the range of degrees of freedom to be fixed. This line may be repeated for all nodes with applied displacements constraints. The three degrees of freedom at each node are denoted by 1,2,3 and correspond to the global (x,y,z) coordinate or the **u**, **v**, and **w** translational motions with respect to the global coordinate system. This is shown in Figure 5. The second statement allows a node set to be used to define fixity conditions.

***BOUNDARY2**
NID, $\mathbf{DOF_1, DOF_2}$

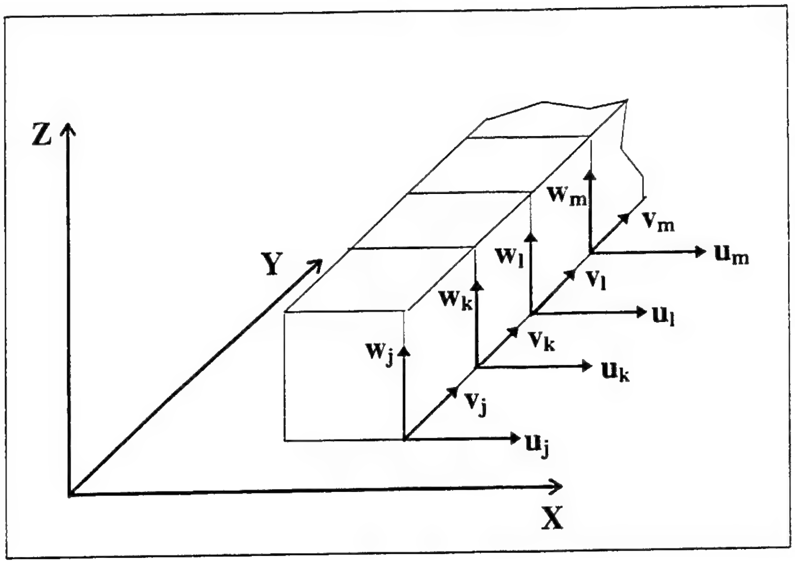


Figure 5. Translational degrees of freedom.

where **NID** is a node set number, and **DOF**$_1$ and **DOF**$_2$ define the range of degrees of freedom to be removed from each node in the set.

External loads may be applied as either concentrated forces at nodes or as pressure distributions over an element face. For concentrated loads the input entry is given by

***CLOAD**
NODE, DOF, MAG

where **NODE** is the node number at the point of application, **DOF** is the direction of the load along the (1,2,3) degree of freedom directions which correspond to the global (x,y,z) coordinate system, and **MAG** is the magnitude of the load. The direction of the load is specified by the sign of the applied force. This line may be repeated for all nodes with applied concentrated loads.

Uniform distributed loading is applied using the ***PRESSURE** statement. This internally integrates the applied pressures and forms a consistent load vector which is included with any other applied point loads. The syntax is given by

***PRESSURE**
ELID, $\mathbf{F}_i$, $\mathbf{P}_o$

where **ELID** is the element number, $\mathbf{F}_i$ is the element face number as shown in Figure 6, and $\mathbf{P}_o$ is the applied pressure. The sign on $\mathbf{P}_o$ dictates whether the distributed tractions are applied opposite to the direction of the outward normal to the face yielding compressive loads ($+\mathbf{P}_o$), or in the same sense as the outer normal yielding tensile loading ($-\mathbf{P}_o$).

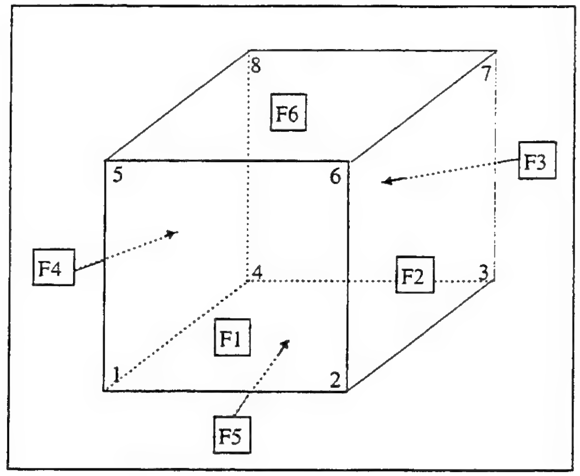


Figure 6. Convention for element face numbering.

2.3.5 Model Generation

It is expected that most models input to RESTRAN will be generated using preprocessing software such as PATRAN, which supports templates for output of a finite element model in various formats. The bulk of an input file typically consists of node and element definitions; for that reason, the adopted form of RESTRAN input syntax for node, element, force, and displacement constraint conditions is based on a simplified form of ABAQUS input formats. Thus, for example, output from a preprocessor in ABAQUS format can be cut and pasted into a RESTRAN input file with only minor modification, and only those statements specific to RESTRAN need be added to complete a job description. Alternatively, the ***INCLUDE** statement can be used to temporarily redirect input to an alternate file containg statements to be used by RESTRAN.

However, in the absence of preprocessing software to facilitate the creation of a finite element model, an input option is available to create a simple layered plate model containing a central circular or elliptical region which may be assigned reduced moduli and/or used to define delamination planes. The use of this rudimentary feature has been assumed within a prepass mode after which the output may be incorporated into a complete RESTRAN input file. This statement is given as follows

***MODEL GENERATION**
NDIV, NSURF, A, B, $\mathbf{ND_1}$, $\mathbf{ND_2}$, $\mathbf{ND_3}$, $\mathbf{ND_4}$
SFCT, $\mathbf{NQ_1}$, $\mathbf{NQ_2}$, $\mathbf{NQ_3}$, $\mathbf{NQ_4}$, NI, NORIENT
$\mathbf{T_{11}, T_{12}, T_{13}}$
$\mathbf{T_{21}, T_{22}, T_{23}}$
$\mathbf{T_{31}, T_{32}, T_{33}}$
1) $\mathbf{SCL_i}$, $\mathbf{ZO_i}$
2) $\mathbf{MATI_i}$, $\mathbf{MATO_i}$, $\mathbf{NSID_i}$, $\mathbf{NOF_i}$, $\mathbf{NEF_i}$

where the various input parameters are identified next and will be discussed subsequently.

NDIV	=	Number of inner divisions
NSURF	=	Number of surfaces
A	=	Length of rectangle
B	=	Width of rectangle
$\mathbf{ND}_i$	=	Number of divisions in outer regions
SFCT	=	Sizing parameter
$\mathbf{NQ}_i$	=	Flag to remove elements in quadrants
NI	=	Flag to remove elements within ellipse
NORIENT	=	ID of orientation vector
$\mathbf{T}_{ij}$	=	Coordinate transformation matrix
$\mathbf{SCL}_i$	=	Scale factor for the i^{th} surface
$\mathbf{Z0}_i$	=	Z-Coordinate of surface
$\mathbf{MATI}_i$	=	Layup number for material properties in inner elliptical region
$\mathbf{MATO}_i$	=	Layup number for material properties in outer elliptical region
$\mathbf{NSID}_i$	=	Node set number for nodes within elliptical region
$\mathbf{NOF}_i$	=	Starting node sequence number
$\mathbf{NEF}_i$	=	Starting element sequence number

The input mixes data for surfaces and layers such as starting node number, **NOF**, and starting element number, **NEF**, respectively. The number of surfaces for each node plane is equal to the number of layers plus one. Element data is assumed to correspond to the layer below the current surface. Therefore, input lines **1** and **2** are repeated for each surface starting at the top of the laminate and, for the last surface, arbitrary values for layer quantities associated with elements, such as material properties, must be entered to avoid null values on the input line but are not used internally.

The model is formed in several stages for each layer. The orientation is initially assumed parallel with the global (x,y,z) coordinates with the z-coordinate defining the thickness direction. First, an inner square region defined on $-1 \leq x, y \leq 1$ with side dimensions equal to two units is meshed containing an inner circular region with a radius fixed at 0.65. The degree of discretization is determined by specifying the number of divisions, **NDIV**, as shown in Figure 7. Next, a larger outer square region is created using corner vertices located at $(\pm\mathbf{SFCT}, \pm\mathbf{SFCT})$, yielding a square with side dimensions $2 \times \mathbf{SFCT}$. As shown in Figure 8, the corner regions are discretized according to the input values $\mathbf{ND}_1$, $\mathbf{ND}_2$, $\mathbf{ND}_3$, $\mathbf{ND}_4$ which specify the number of divisions to be made in each region. For each surface, the scale factor $\mathbf{SCL}_i$ may be used to shrink or expand the size of the delamination through the thickness. This value is applied to the node coordinates within the inner region such that $(x', y') = \mathbf{SCL}_i(x, y)$ for $-1 \leq (x, y) \leq 1$. Acceptable values for this magnification factor should be positive and less than or equal to unity. Figure 9 shows an enlarged side view of a four-layered plate with a varying delamination radius created by using **SCL** = 1.0, 0.9, 0.8, 0.7 and 0.6 in the five different surfaces. The relative size of the inner square to the outer square region can be varied through the selection of **SFCT** and **SCL**, and is given by the ratio **SCL/SFCT**. A final transformation is performed wherein the coordinates of the entire node field is multiplied by

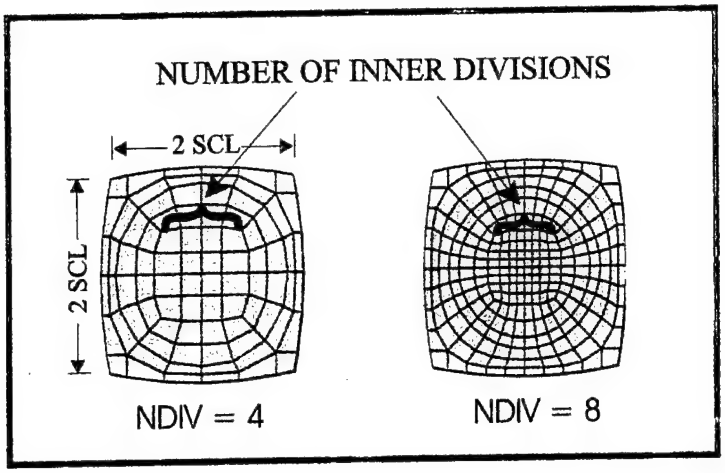


Figure 7. Controlling the level of discretization using the **NDIV** parameter.

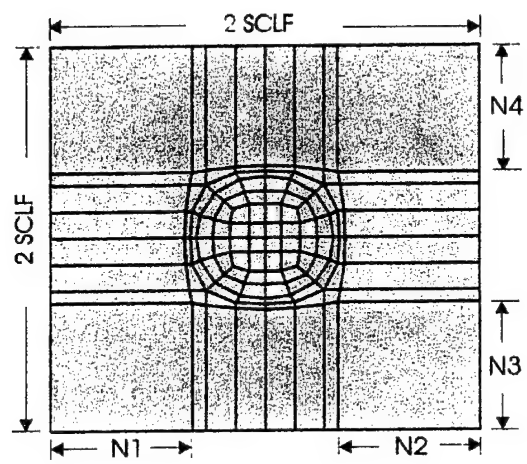


Figure 8. Discretization of outer corner regions.

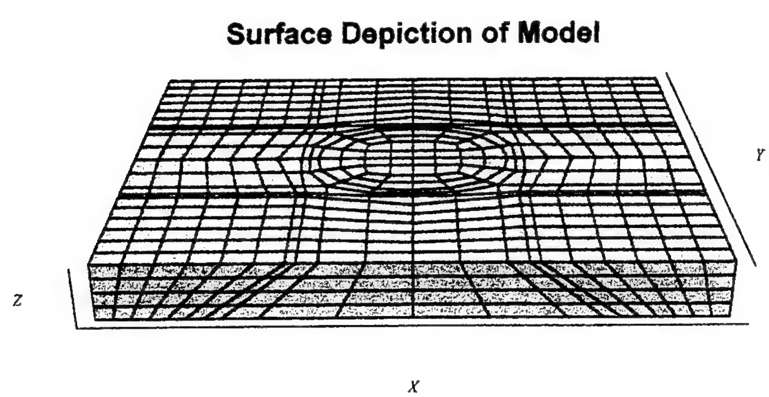


Figure 9. Variable delamination size through thickness.

the factor 1/**SFCT** and then scaled according to the stretch factors **A** and **B** to give the desired overall plate dimensions. For **A** = **B**, the inner region remains circular, while for **A** $\neq$ **B**, different elliptical regions may be generated. As shown in Figure 10, various input values for **SFCT**, **SCL**, **A**, and **B** can yield significantly different models.

The starting numbers for nodes in each surface and elements in each layer must be specified such that the last generated node in a surface or element number in a layer does not exceed the starting number assigned to the node or element in the next surface or layer. For each surface, a node set number, **NSID**, is input to identify the nodes within the in inner circular region. These nodes are automatically stored as a node set and can be used to define an inner delamination node plane. For each layer, material property identification numbers are given for elements within and outside the elliptical region in the **MATI** and **MATO** parameters, respectively.

If symmetry can be used to reduce the size of the model, quadrants may be selected for removal using the flags $\mathbf{NQ}_1$, $\mathbf{NQ}_2$, $\mathbf{NQ}_3$, $\mathbf{NQ}_4$ in which a value of zero causes all elements and nodes in the corresponding quadrant to be deleted. The designation for the quadrants and models exploiting various symmetry conditions obtained using different values for the flags $\mathbf{NQ}_i$ is shown in Figure 11. To create a model with an open hole, setting the modeling flag **NI** equal to zero will cause all elements within the ellipse to be removed.

Finally, the coordinate transformation matrix is used to orient the model in space. This matrix maps the coordinates as

$$\left\{ \begin{array}{c} \bar{x} \\ \bar{y} \\ \bar{z} \end{array} \right\} = \begin{bmatrix} t_{11} & t_{12} & t_{13} \\ t_{21} & t_{22} & t_{23} \\ t_{31} & t_{32} & t_{33} \end{bmatrix} \left\{ \begin{array}{c} x \\ y \\ z \end{array} \right\} \tag{3}$$

Figure 12 shows an example with a transformation matrix given by

$$[T] = \begin{bmatrix} 0.0 & 0.0 & 1.0 \\ 0.0 & 1.0 & 0.0 \\ 1.0 & 0.0 & 0.0 \end{bmatrix} \tag{4}$$

together with the modeling option flag **NI** = 0 which creates a layered model of a vertical plate with an open elliptical hole.

This routine has been included to create a simple model and was designed to be used in a ***PREPASS** execution mode with graphics outputted to show the generated model. The generated elements and nodes are output to the file *restran.elg*, which is formatted to list elements and nodes together with nodes identified along the various boundaries. This format is shown in Figure 13. Upon completion, the node and element definitions that are generated may be subsequently included into a new input file with all loads and boundary conditions specified in order to perform an analysis.

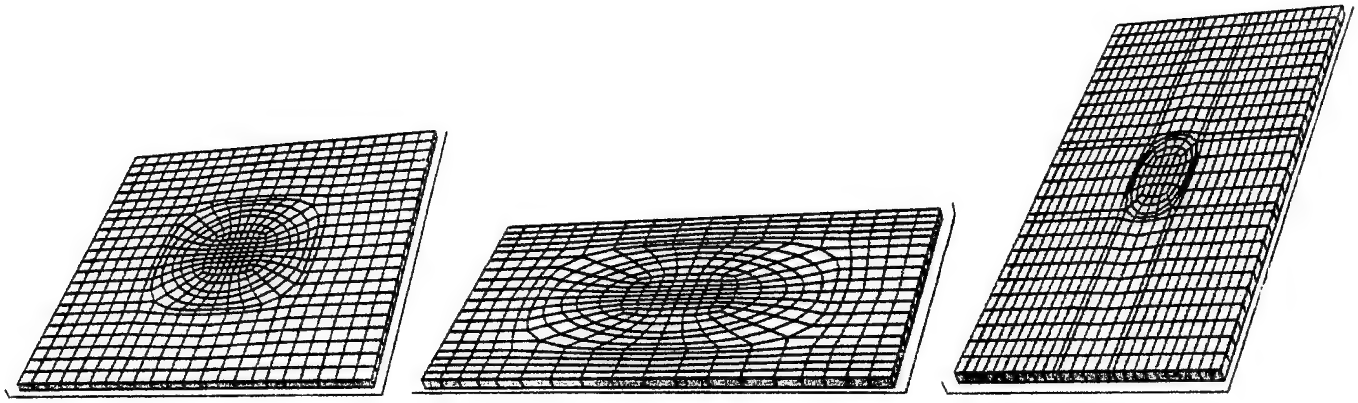


Figure 10. Relative model sizes.

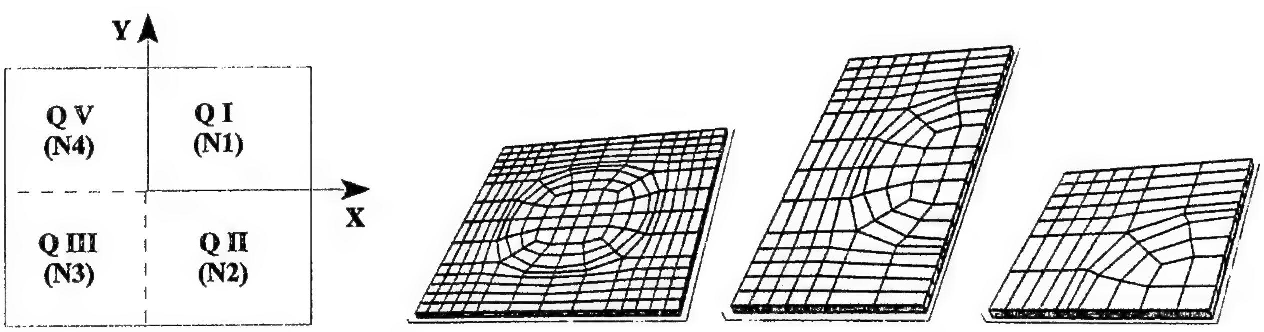


Figure 11. Selection of quadrants to model symmetry conditions.

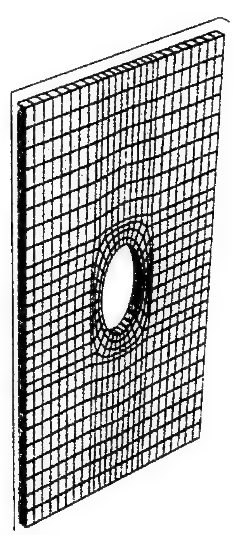


Figure 12. Rotated plate in space with an open elliptical hole.

```
*NODE
       1  0.0000000E+00  0.0000000E+00  0.3000000E+01
       2  0.0000000E+00  0.1000000E+01  0.3000000E+01
       3  0.2000000E+01  0.1000000E+01  0.3000000E+01
       4  0.2000000E+01  0.0000000E+00  0.3000000E+01
       5  0.2000000E+01 -0.1000000E+01  0.3000000E+01
       6  0.0000000E+00 -0.1000000E+01  0.3000000E+01
      10  0.0000000E+00  0.2666667E+01  0.3000000E+01

*NSET, NSID =      55
   1002    1003    1004    1005    1006    1007    1008    1009
   1010    1011    1012    1013    1014    1015    1016    1017
   1018    1019    1020    1021    1022    1023    1024

*NSET, NSID =      60
   2002    2003    2004    2005    2006    2007    2008    2009
   2010    2011    2012    2013    2014    2015    2016    2017
   2018    2019    2020    2021    2022    2023    2024

*NSET, NSID =      65
   3002    3003    3004    3005    3006    3007    3008    3009
   3010    3011    3012    3013    3014    3015    3016    3017
   3018    3019    3020    3021    3022    3023    3024

*NSET, NSID =      70
   4002    4003    4004    4005    4006    4007    4008    4009
   4010    4011    4012    4013    4014    4015    4016    4017
   4018    4019    4020    4021    4022    4023    4024

*ELEMENT, LAYUP =         1
  1    1001    1004    1003    1002       1       4       3       2
  2    1006    1005    1004    1001       6       5       4       1
  5    1002    1003    1011    1010       2       3      11      10
  6    1003    1013    1012    1011       3      13      12      11
  7    1004    1014    1013    1003       4      14      13       3
  8    1005    1015    1014    1004       5      15      14       4
  9    1017    1016    1015    1005      17      16      15       5
 10    1018    1017    1005    1006      18      17       5       6

*ELEMENT, LAYUP =         2
 65    1058    1059    1075    1074      58      59      75      74
 66    1059    1060    1076    1075      59      60      76      75
 67    1060    1078    1077    1076      60      78      77      76
 68    1061    1079    1078    1060      61      79      78      60
 69    1062    1080    1079    1061      62      80      79      61
 70    1063    1081    1080    1062      63      81      80      62
 71    1064    1082    1081    1063      64      82      81      63

               NODES CONTAINED ON OUTER SIDE BOUNDARIES

      NODES ALONG (0.000E+00,0.100E+02) <-> (0.200E+02,0.100E+02)
```

Figure 13. Sample format of file *restran.elg*.

```
 266   267   268   269   270   271   272   273  1266  1267
1268  1269  1270  1271  1272  1273  2266  2267  2268  2269
2270  2271  2272  2273  3266  3267  3268  3269  3270  3271

      NODES ALONG (0.000E+00,-.100E+02) <-> (0.000E+00,0.100E+02)

   2     6    10    18    26    34    42    50    58    66
  74    86   105   120   135   150   221   236   251   266
1001  1002  1006  1010  1018  1026  1034  1042  1050  1058

      NODES ALONG (0.200E+02,-.100E+02) <-> (0.200E+02,0.100E+02)

 112   127   142   157   165   173   181   189   197   205
 213   228   243   258   273  1112  1127  1142  1157  1165
1173  1181  1189  1197  1205  1213  1228  1243  1258  1273
2112  2127  2142  2157  2165  2173  2181  2189  2197  2205
```

Figure 13. Sample format of file *restran.elg* (continued).

2.3.6 Material Property Input

The complete specification of material properties encompasses the description of lamination layup, elastic moduli, failure criteria, initial material damage, and property degradation laws. The input of these quantities is described in the following subsections.

2.3.6.1 Linear Elastic Material Moduli

The assumption of a layered material media in RESTRAN requires that a sequence of plies be specified even in the case of a homogeneous material. This is inputted through the following statement

***LAYER, LAYUP = M**
MATID$_i$, THK$_i$, THETA$_i$

where the mandatory parameter **LAYUP = M** designates the layer identification number which is matched to the **LAYUP = M** parameter on the ***ELEMENT** data entry. **MATID$_i$** is the identification number of a ***MATERIAL** statement which gives the material moduli for the i^{th} ply. The **THK$_i$** and **THETA$_i$** entries give the layer thickness and ply orientation angle with respect to the global or local coordinate system, respectively. This data line is repeated for each ply in the sublaminate.

Each layer may assume different orthotropic material properties with nine independent material moduli. These moduli are input using the following statement:

***MATERIAL, MATID = M$_i$**
E$_1$, E$_2$, E$_3$, G$_{13}$, G$_{23}$, G$_{12}$
V$_{13}$, V$_{23}$, V$_{12}$

where the subscripts (1,2,3) refer to the principle fiber coordinates for composite materials or

general material axes for homogeneous orthotropic materials.

2.3.6.2 Nonlinear Elastic Material Moduli

The support of nonlinear material behavior in RESTRAN is restricted to materials which exhibit a nonlinear-elastic stress-strain relationship. For residual strength characterization, nonlinear inelastic behavior such as plastic deformation is considered a failure mechanism and may be accounted through selected failure criteria and damage laws. Nonlinear elastic materials, which follow the same deformation path under loading and unloading as shown in Figure 14, are indicated through the following statement.

***NONLINEAR MATERIAL**
ITLIM, CVMTOL, CVBTOL
$\mathbf{P}_1$,$\mathbf{P}_2$,$\mathbf{P}_3$, ... ,$\mathbf{P}_N$

where **ITLIM** is the maximum number of iterations permitted to obtain a converged solution. **CVMTOL** is the tolerance of the difference in the norm of the residual load vector between iterations below which convergence of material failure is determined.

$$\|\mathbf{\Delta R}_i\| \leq \mathbf{CVMTOL} \tag{5}$$

CVBTOL is the corresponding prescribed tolerance for predicting convergence of buckling eigenvalues such that

$$\|\lambda_\mathbf{i} - \lambda_\mathbf{i-1}\| \leq \mathbf{CVBTOL} \tag{6}$$

The ***NONLINEAR MATERIAL** statement additionally alerts RESTRAN to treat material properties specified on the ***MATERIAL** entry as initial values and to subsequently access a user-defined subroutine denoted **USERNL** to obtain nonlinear moduli under increasing applied loads. A user-defined subroutine mode of input was selected as the most general and straightforward method of defining nonlinear elastic properties. As opposed to tabular entry formats, **USERNL** allows all material moduli to be varied independently provided that the resulting constitutive matrix remains positive definite. Also, specific moduli may be written as a function of individual stress or strain components, or based on combined measures of unit elastic energy or strain energy density. RESTRAN passes interpolated stress and strain data at each ply transformed into the local material coordinate system of the current element and requires that the user routine return scale factors to be applied to the initial moduli to approximate secant material properties. The input of the number of iterations and convergence tolerance parameters is optional, and default values of **ITLIM** = 5, **CVMTOL** = 0.01, and **CVBTOL** = 2 are internally assigned. The constants $\mathbf{P}_N$ are optional input parameters that are passed into the user-defined subroutine and may be used to define secant properties. A maximum of 25 parameters may be input on as many additional lines as needed.

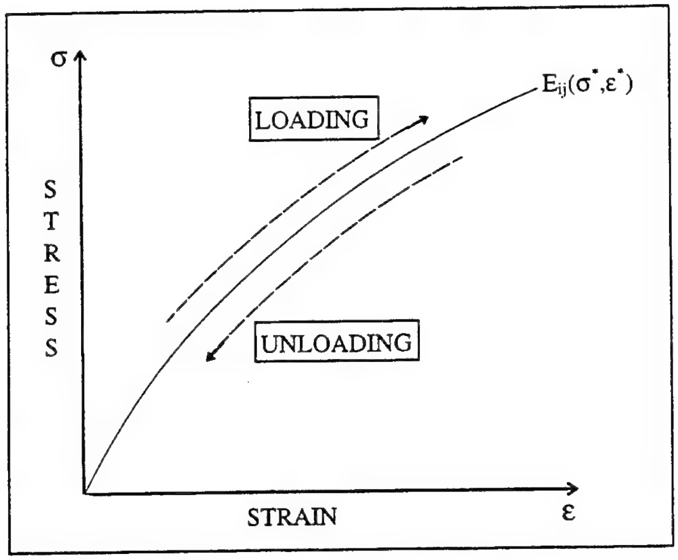


Figure 14. Nonlinear elastic material.

The subroutine interface written in FORTRAN is described next. The current implementation in RESTRAN assumes a single nonlinear material that can be described by a maximum of 25 parameters specified in the input file (any number can be 'built-in' to the user written subroutine). With proper consideration of the argument call interface, this and all user-defined subroutines may be developed in selected languages, compiled, and linked with the main RESTRAN executable prior to job execution. Figure 15 shows the required subroutine format.

```
      SUBROUTINE USERNL ( STRESS,STRAIN,SXX,SYY,SZZ,TYZ,TZX,TXY,EXX,EYY,
     1                    EZZ,GYZ,GZX,GXY,PARAM,THKN,ATHK,TLM,THETA,FE1,
     2                    FE1,FE2,FE3,FG31,FG23,FG12,FV31,FV23,FV12,NORD )
C
C     ************************************************
C     **                                            **
C     **   USER-DEFINED ROUTINE TO COMPUTE SECANT   **
C     **   MODULI IN NONLINEAR MATERIAL ANALYSIS    **
C     **                                            **
C     ************************************************
C
      IMPLICIT DOUBLE PRECISION (A-H,O-Z)
C
      DIMENSION STRESS(27,6),STRAIN(27,6),PARAM(25)
C
      *** USER DEFINED CODE FOR COMPUTING ***
      *** SECANT MATERIAL MODULI          ***
C
      RETURN
      END
```

Figure 15. **USERNL** subroutine format.

This routine is called for each layer in each element for the calculation of material properties as a function of stress/strain state. Stresses and strains are passed to the user subroutine calculated at Gauss points and at the midpoint of each element layer rotated into the local coordinate system if one has been specified. Gauss-point tensor quantities are stored in the STRESS and STRAIN arrays for which the storage order depicted in Figure 16 is the same for both stresses and strains. Interpolated layer stresses are provided in the scalar variables SXX $\rightarrow$ TXY, while corresponding strain values are stored in EXX $\rightarrow$ GXY. The total laminate and layer thicknesses are given by the TLM and ATHK variables. The accumulated thickness to the current ply is stored in the THKN variable. THETA contains the orientation angle of the current ply. PARAM is an array containing input parameters to define the material nonlinearity. The user routine must return scalar factors FE1 $\rightarrow$ FV12 which RESTRAN uses to compute secant moduli as depicted in Figure 17 for the current load level.

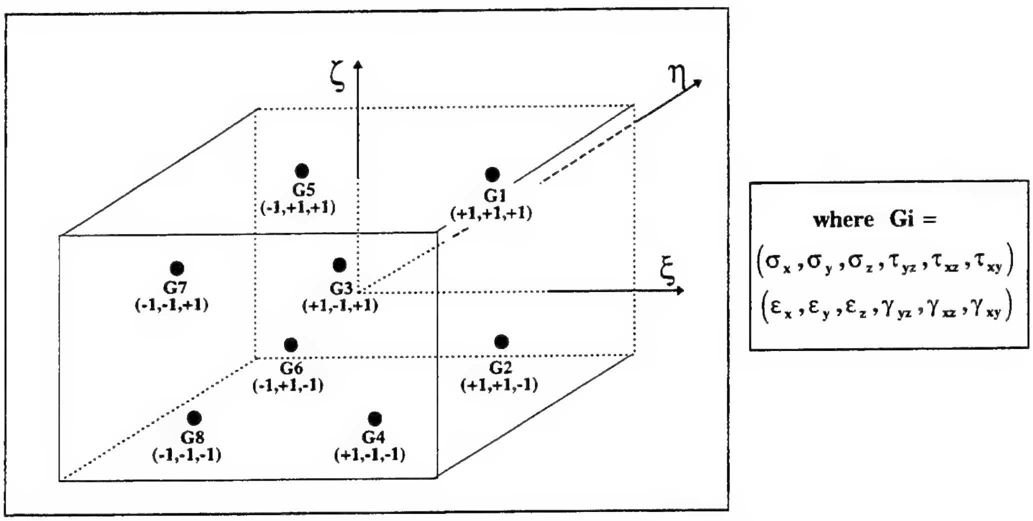


Figure 16. Storage format for stresses and strains.

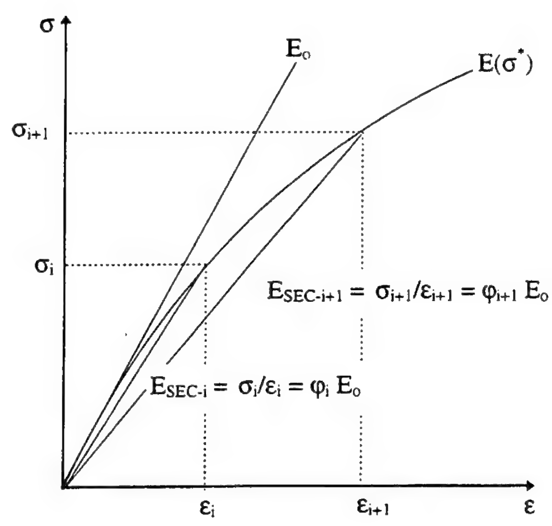


Figure 17. Calculation of secant moduli.

These factors are applied to the initial moduli as

$$\begin{aligned}
\bar{E}_1^k &= F_{E1}\ E_{1o}^k & \bar{\nu}_{31}^k &= F_{V31}\ \nu_{31o}^k \\
\bar{E}_2^k &= F_{E2}\ E_{2o}^k & \bar{\nu}_{23}^k &= F_{V23}\ \nu_{23o}^k \\
\bar{E}_3^k &= F_{E3}\ E_{3o}^k & \bar{\nu}_{12}^k &= F_{V12}\ \nu_{12o}^k \\
\bar{G}_{23}^k &= F_{G23}\ G_{23o}^k \\
\bar{G}_{31}^k &= F_{G31}\ G_{31o}^k \\
\bar{G}_{12}^k &= F_{G12}\ G_{12o}^k
\end{aligned} \tag{7}$$

This user-defined subroutine must be compiled and linked into the RESTRAN executable prior to performing residual strength analysis. An example of the user-written subroutine **USERNL** for returning secant moduli is presented in Appendix A. Finally, it must be noted that if increased execution time in converging nonlinear material response is problematic or nonlinear properties are slight or inaccurately characterized, the nonlinear solution algorithm and the use of this subroutine may be avoided by inputting initial moduli as effective secant approximations based on measured ultimate failure strength and strain data as depicted in Figure 18.

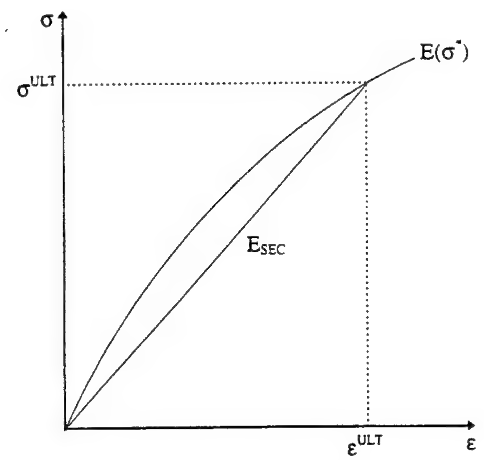


Figure 18. Linear approximation to nonlinear elastic moduli.

2.3.7 Failure Mode Selection

For residual strength prediction, the selection of the **METH** parameter on the ***SOLUTION** entry determines whether material failure modes and/or buckling failure modes are to be calculated in the algorithmic assessment of progressive failure.

2.3.7.1 Material Failure Criteria

RESTRAN supports a robust suite of material failure criteria. The specific failure criterion to be associated with an input material labeled **Mi** is specified using

> ***FAILURE CRITERION, MATID = Mi**
> **FCTYPE**
> $\mathbf{P_1, P_2, P_3, P_4, P_5, P_6}$
> $\mathbf{P_7, P_8, P_9, P_{10}}$

where **FCTYPE** is a character string which denotes the name of the criterion, and $\mathbf{P}_1$ through $\mathbf{P}_{10}$ are used to input up to 10 material constants required by the criterion. For the various input parameters, compressive material strengths may be input as positive or negative, the sign is internally corrected. The criteria designations and required constants are listed in Table 2.

Table 2: Parameters required for various failure criteria.

Criterion	Parameters
MAX-STRESS	X_t, X_c, Y_t, Y_c, Z_t, Z_c, R, S, T
MAX-STRAIN	X_t, X_c, Y_t, Y_c, Z_t, Z_c, R, S, T
BELTRAMI	S^{ult}
VON MISES	S^{ult}
HOFFMAN(1,2,3)	X_t, X_c, Y_t, Y_c, Z_t, Z_c, R, S, T
HILL(1,2,3)	X_t, X_c, Y_t, Y_c, Z_t, Z_c, R, S, T
TSAI-WU(1,2,3)	X_t, X_c, Y_t, Y_c, Z_t, Z_c, R, S, T
CHRISTENSEN	X_t, X_c, K, α
FENG	A_1, A_2, A_3, A_4, A_5, A_6
HASHIN	X_t, X_c, Y_t, Y_c, S
USER-DEFINED	NRCHK, $P_1 \rightarrow P_{10}$

A complete description of the various failure criteria, including the required material constants, is presented in the RESTRAN theoretical manual [3]. In summary, X_t, X_c, Y_t, Y_c, Z_t, and Z_c are the normal tensile and compressive strengths in the principle 1, 2, and 3 directions, respectively, and R, S, and T are the shear strengths defined in the 23, 13, and 12 planes, respectively. For the isotropic Beltrami and Von Mises criteria, S^{ult} is the combined stress measure at which yielding occurs or the proportional limit reached. The constants required in the Christensen, Feng, and Hashin criteria are explained in the RESTRAN Theoretical Manual [3]. The variations on the Hoffman, Hill, and Tsai-Wu criteria involve specialization of these criteria to account for two-dimensional or three-dimensional stress states and whether they predict a single unspecified failure mode or differentiate between fiber and matrix modes. The variations are specified by the appended numerical suffix in which '1' designates 3-D single mode failure, '2' designates 2-D plane stress, single mode failure, and '3' specifies 3-D mixed mode failure. The user-defined option is discussed in the following subsection.

2.3.7.2 User-Defined Failure Criteria

Specifying a user-defined failure criteria alerts RESTRAN to access a user-supplied subroutine designated USERFC. The output from this subroutine is an unaltered or degraded set of material properties for the current ply depending on whether the user-defined criterion has predicted material failure. The input parameter NRCHK controls error checking in a subroutine driver for USERFC. If NRCHK is equal to zero, all checks will be bypassed. A nonzero entry will cause the following tests to be performed:

1) If moduli or Poisson ratios have increased, a warning will be issued and execution will con-

tinue.

2) If moduli have been set to negative values, an error message will be printed and execution will continue with moduli set equal to zero.

3) If scale factors are returned less than or equal to zero, an error message will be printed and an absolute value of the scale factor will be used in subsequent calculations.

The parameters $P_1 \rightarrow P_{10}$ are optional and may be used to pass required data into the user-defined subroutine. The format of USERFC is shown in Figure 19.

```
      SUBROUTINE USERFC ( SGPTS,EGPTS,STRESS,STRAIN,PTHK,THETA,IPLY,
     1                    NELID,PARAM,E1,E2,E3,G12,G23,G31,V12,V23,
     2                    V13,EM,EF,NORD,NSTAT,NFSTAT,NACCS )
C
C     *************************************************
C     **                                             **
C     **  USER-DEFINED FAILURE CRITERION TO COMPUTE  **
C     **  MATERIAL FAILURE UNDER APPLIED LOAD        **
C     **                                             **
C     *************************************************
C
      IMPLICIT DOUBLE PRECISION (A-H,O-Z)
C
      DIMENSION SGPTS(27,6),EGPTS(27,6),STRESS(6),STRAIN(6),PARAM(15)
C
      *** SOURCE CODE FOR USER-DEFINED FAILURE ***
      *** CRITERION AND DAMAGE LAW             ***
C
      RETURN
      END
```

Figure 19. **USERFC** subroutine format.

The input and output data passed into the routine through the argument list will be explained subsequently. STRESS, and STRAIN are vectors of stress and strain interpolated to the current ply and transformed into the local material coordinate system. The storage convention for the stress/strain vectors is given by $[E_{11}, E_{22}, E_{33}, E_{23}, E_{31}, E_{12}]$. SGPTS and EGPTS are arrays which store the untransformed stresses and strain components at the element integration or Gauss points. PTHK and THETA are ply thickness and orientation angle, respectively. IPLY and NELID are ply number and element number, respectively. The PARAM array contains a list of parameters from the *FAILURE CRITERIA entry. $E_1 - \nu_{31}$ are material moduli and Poisson ratios. EM, and EF are scale factors for matrix- and fiber-dominated failure modes. NSTAT is the ply status flag in which NSTAT = 1 or 2 for matrix- or fiber-dominated failure mode, respectively, and NSTAT = 3 for total ply failure. In addition, the option exists to set NSTAT = 4, which causes the element to be excluded from further degradation. The new failure status is returned in the variable NFSTAT.

NACCS is set to zero if no failure is predicted during the current iteration and to unity if failure is predicted. An example of a user-written subroutine for determining ply failure indices is presented in Appendix B.

This user-defined subroutine must be compiled and linked into the RESTRAN executable prior to performing residual strength analysis.

2.3.8 Material Damage Laws

Independent of the selected failure criterion, several types of material damage laws may be invoked to degrade material properties. These damage laws are available to selectively apply varying degrees of material degradation to simulate progressive ply damage modes. For complete generality, a user-written subroutine may be used.

2.3.8.1 Damage Law Selection

A specific damage law is selected using the following statement.

> ***DAMAGE LAW, MATID = Mi**
> **APPLIED-DAMAGE-LAW**
> **$\mathbf{P_1, P_2, P_3, P_4, P_5, P_6}$**

where **MATID = $\mathbf{M}_i$** associates the damage law with a material property set identification number. The following parameter, **APPLIED-DAMAGE-LAW**, may be given as '**NULL**', '**MIXED**', or '**PARTIAL**' to invoke specific damage laws. Information regarding failure mode determination by the selected failure criterion is stored and used in applying the selected damage law. The parameters, $\mathbf{P_1}$, $\mathbf{P_2}$, $\mathbf{P_3}$, $\mathbf{P_4}$, $\mathbf{P_5}$, $\mathbf{P_6}$, may be used in the computing degraded material properties. Currently, only $\mathbf{P_1}$ is used when specifying a '**PARTIAL**' degradation in properties. The '**NULL**' option causes all properties of the current ply to be set to zero as

$$\begin{aligned} E_1 &= 0.0 & G_{31} &= 0.0 & \nu_{12} &= 0.0 \\ E_2 &= 0.0 & G_{23} &= 0.0 & \nu_{13} &= 0.0 \\ E_3 &= 0.0 & G_{12} &= 0.0 & \nu_{23} &= 0.0 \end{aligned} \tag{8}$$

When selecting a single-mode failure criteria, specifying '**PARTIAL**' will cause properties to first be degraded according to

$$\begin{aligned} E_1 &= \alpha E_1 & G_{31} &= \alpha G_{31} & \nu_{12} &= \alpha\nu_{12} \\ E_2 &= \alpha E_2 & G_{23} &= \alpha G_{23} & \nu_{13} &= \alpha\nu_{13} \\ E_3 &= \alpha E_3 & G_{12} &= \alpha G_{12} & \nu_{23} &= \alpha\nu_{23} \end{aligned} \tag{9}$$

For differentiating between fiber and matrix failure modes, the options **MIXED** or **PARTIAL** may be selected. Both degrade selected material properties depending on predicted failure mode. For a partial degradation, a reduction factor, α, is entered through the $\mathbf{P}_1$ parameter. In specifying a mixed mode degradation, the factor α is automatically taken as zero. The degradation laws are given by the following.

For fiber failure mode

$$\begin{aligned}
E_1 &= \alpha E_1 \\
G_{31} &= \alpha G_{31} \\
G_{12} &= \alpha G_{12} \\
\nu_{12} &= \alpha \nu_{12} \\
\nu_{13} &= \alpha \nu_{31}
\end{aligned} \tag{10}$$

For matrix dominated failure

$$\begin{aligned}
E_2 &= \alpha E_2 \\
E_3 &= \alpha E_3 \\
G_{23} &= \alpha G_{23} \\
\nu_{23} &= \alpha \nu_{23}
\end{aligned} \tag{11}$$

For interlaminar shearing mode failure

$$\begin{aligned}
G_{23} = \alpha G_{23} \\
G_{31} = \alpha G_{31}
\end{aligned} \tag{12}$$

For inplane shearing failure mode

$$G_{12} = \alpha G_{12} \tag{13}$$

After initial partial ply failure is accounted, subsequent failure is assumed total.

2.3.8.2 User-Defined Damage Law

A user-defined damage law is applied through selecting the **USER-DEFINED** failure criterion on the ***FAILURE CRITERIA** entry. The USERFC subroutine detailed in Figure 19 is then accessed to apply the specific failure criteria and return reduced material properties using the user-defined damage law encoded into this subroutine.

2.3.9 Initial Damage Description

RESTRAN has been specifically designed to predict residual strength of composite laminates which contain initial damage due to impact events. The input of the initial damage state is critical in the prediction of residual strength and is, perhaps, the most difficult data to obtain from a user's standpoint. The following section describes the input requirements.

2.3.9.1 Initial Material Damage

The input of initial damage in RESTRAN is performed using the following data entry:

> ***INITIAL DAMAGE**
> **ELSET, DAMAGE-TYPE**
> $\mathbf{P_1, P_2, \ldots, P_{10}}$

where **ELSET** specifies an element set to which the initial damage is to be applied. Damage is assumed to apply equally to all plies in the element. Initial property degradation may be applied

by setting the the **DAMAGE-TYPE** parameter equal to '**FIBER**', '**MATRIX**', '**TOTAL**', '**USER-DEFINED**', or '**EXCLUDED**' to specify the initial effects of various damage modes. For fiber failure, the following properties are degraded:

$$\begin{aligned} E_1 &= 0 \\ G_{31} &= 0 \\ G_{12} &= 0 \\ \nu_{13} &= 0 \\ \nu_{12} &= 0 \end{aligned} \tag{14}$$

For matrix failure, the following properties are degraded:

$$\begin{aligned} E_2 &= 0 \\ E_3 &= 0 \\ G_{23} &= 0 \\ \nu_{23} &= 0 \end{aligned}$$

For total failure, all material properties are set equal to zero such that

$$\begin{array}{lllllllll} E_1 & = & 0.0 & G_{31} & = & 0.0 & \nu_{12} & = & 0.0 \\ E_2 & = & 0.0 & G_{23} & = & 0.0 & \nu_{13} & = & 0.0 \\ E_3 & = & 0.0 & G_{12} & = & 0.0 & \nu_{23} & = & 0.0 \end{array} \tag{15}$$

Specifying '**USER-DEFINED**' on the ***INITIAL DAMAGE** entry, the input parameters, $\mathbf{P}_1 \rightarrow \mathbf{P}_{10}$, are passed into the user-defined subroutine USERID for initial calculation of reduced material properties to simulate initial damage. The format of USERID is shown in Figure 20.

```
      SUBROUTINE USERID (PTHK,THETA,E1,E2,E3,G13,G23,G12,V13,V23,V12,
     1                   P1,P2,P3,P4,P5,P6,P7,P8,P9,NELID,IPLY,NSTAT )
C
C     **************************************************
C     **                                              **
C     **     USER-DEFINED SUBROUTINE TO ASSIGN        **
C     **     INITIAL ELEMENT MATERIAL DAMAGE.         **
C     **                                              **
C     **************************************************
C
      IMPLICIT DOUBLE PRECISION (A-H,O-Z)
C
      *** USER-DEFINED INITIAL MATERIAL ***
      *** DAMAGE SOURCE CODE            ***
C
      RETURN
      END
```

Figure 20. **USERID** subroutine format.

PTHK and THETA are the current ply thickness and orientation angle, E1 through V12 are material moduli, P1 through P9 are input parameters provided on the ***MDAMAGE** data entry, and NELID and IPLY are the current element and ply number. NSTAT is a ply failure status flag which is set to '1' or '2' for matrix or fiber (or general) failure, respectively. This permits additional material failure to be predicted. If total failure is predicted, NSTAT is set equal to '3'. This is usually assigned when the material moduli have been reduced to zero. If NSTAT is set equal to '4,' then the current damage state is maintained and further degradation is excluded. This may be enforced initially through a damage-type specification discussed next. An example of a user-written subroutine to assign initial material damage is contained in Appendix C.

The specification of '**EXCLUDED**' is provided to indicate that the current element set is free of initial damage and is excluded from accumulating damage during subsequent residual strength analysis. This feature permits, in perhaps the easiest manner, an arbitrary set of elements to be declared failure free or indestructible. This option may be used to prevent elements from failing at which external loads are applied - which will normally terminate the analysis - or to avoid failing some internal structure which is deemed not to influence residual strength predictions. The use of this method for protecting elements or by the parameter **LAPFAIL** is advocated over defining a set of elements with high material moduli or strength measures because these approaches could, in uncommon situations, cause these exaggerated element properties to be used in predicting ultimate failure, thus yielding incorrect measures of residual strength.

2.3.9.2 Delamination Description

A delamination is defined by a node set which identifies the node points lying within the delamination plane. RESTRAN automatically generates a corresponding set of coincident nodes and redefines element connectivity to sever connection between elements lying above and below this plane defined by the normal to the delamination surface. The input statement used to define delamination planes is given by

***DELAMINATION**
$\mathbf{NSET_1}$, $\mathbf{NSET_2}$, $\mathbf{NSET_3}$, ... ,$\mathbf{NSET_n}$

where $\mathbf{NSET_i}$ lists node set identification numbers defined in the ***NODE SET** entry. Any number of node sets may be listed per line, and this line may be repeated as many times as needed to define the delaminations present in the model.

RESTRAN currently calculates buckling loads based on a linear stability analysis. To approximate structural behavior in the post-buckled regime, modifications to the material properties of those elements participating in the buckling mode must be specified. A full description of allowable post-buckled element response is discussed in the RESTRAN Theoretical Manual [3]. The following statement is mandatory for residual strength analysis which includes potential sublaminate buckling failure. The entry statement is given by

***POSTBUCKLED MATERIAL**
CLFAC, PBFAC, NITER

where **CLFAC** specifies whether a constant load at buckling failure is to be approximately maintained in the buckled elements during the additional progressive failure analysis. This parameter

is a real number ranging from 0.0 to 1.0. For **CLFAC** = 0.0, no internal scaling is performed to approximate the constant load state, and for values greater than 0.0 up to 1.0, varying degrees of maintaining the internal element loads present at failure are enforced. The parameter **PBFAC** is used to reduce element material properties to account for buckling failure. This factor is used to reduce the constitutive properties as

$$[\bar{\mathbf{C}}_{ij}] = \mathbf{PBFAC}[\mathbf{C}_{ij}] \tag{16}$$

Both **CLFAC** and **PBMFAC** may be specified to account for assumed post-buckled sublaminate behavior. For **CLFAC** > 0, the global stiffness matrix becomes a nonlinear function of loads at which prior sublaminates have undergone buckling failure, and an iterative procedure is invoked to converge subsequent eigen-analyses. The parameter **NITER** is used to specify the maximum number of iterations to be performed in calculating further sublaminate buckling loads.

2.3.10 Coordinate Transformation

An arbitrary coordinate system $(\bar{x}, \bar{y}, \bar{z})$ may be specified for each element in RESTRAN to define a local reference system. This local coordinate system may be used to define ply material properties and account for any laminate orientation with respect to the element (x',y',z') system which is aligned with the global coordinate system. On output, element or layer stresses and strains may be output in this local system. These Cartesian coordinate systems are depicted in Figure 21.

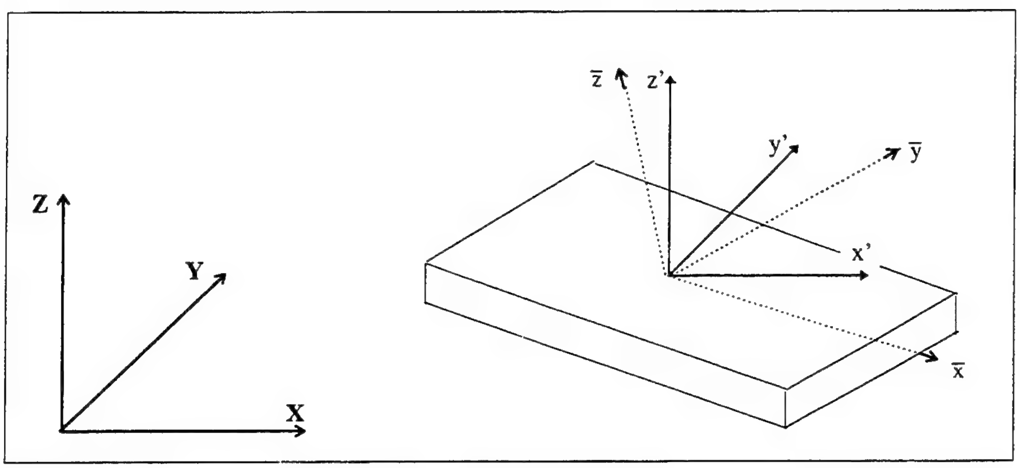


Figure 21. Global (x,y,z), element (x',y',z') and local $(\bar{x}, \bar{y}, \bar{z})$ coordinate systems.

A local coordinate system is defined using the following input statement

***ORIENTATION**
CID, $\mathbf{Vx_1}$, $\mathbf{Vx_2}$, $\mathbf{Vx_3}$, $\mathbf{Vy_1}$, $\mathbf{Vy_2}$, $\mathbf{Vy_3}$

where **CID** is the coordinate system number, and $\mathbf{Vx}_1 \rightarrow \mathbf{Vy}_3$ are the components of two vectors defined in the global coordinate system that define a plane. If these two vectors are not orthogonal, the **Vy** vector is made orthogonal to **Vx** and the third coordinate **Vz** is determined from the vector cross product $\mathbf{Vz} = \mathbf{Vx} \times \mathbf{Vy}$. The coordinate system is associated with selected elements using the **ORIENTATION** parameter on the ***ELEMENT** data entry. The generation of a local coordinate system is depicted in Figure 22.

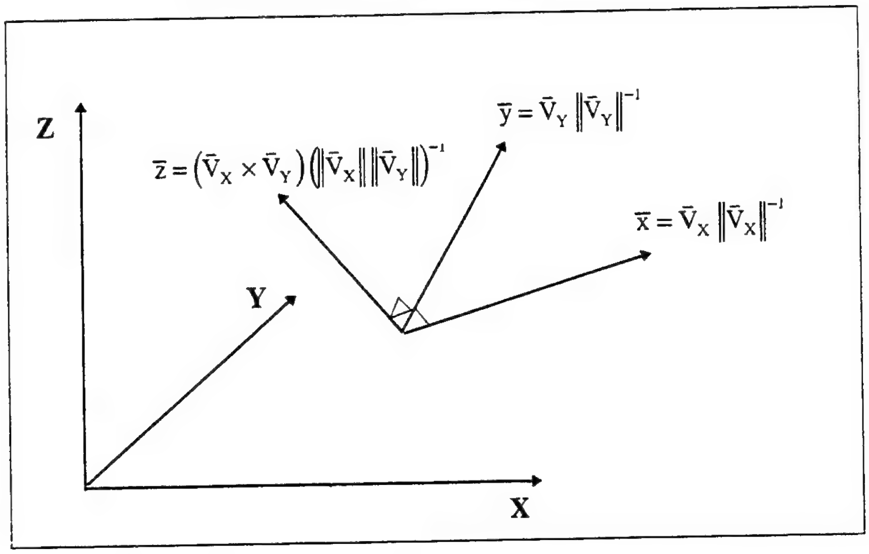


Figure 22. Local coordinate system generation.

To set a tolerance for deformed geometry check, the following statement is required:

***DEFORMED GEOMETRY**
TOL

where **TOL** is the angular tolerance in degrees such that warnings will be issued if the calculated internal element angles, θ, are outside the bounds

$$\mathbf{TOL} \leq \theta \leq 180 - \mathbf{TOL} \tag{17}$$

The default value for the tolerance is 45^o. This will only alert the user to possibly incorrect element definitions or excessive element distortions but will not stop execution.

Nodes may be equivalenced through the following command

***EQUIVALENCE**
NSET1, NSET2

where **NSET1** and **NSET2** are node set identifiers. Any number of node set pairs may be entered on separate lines and must be of equal size. This command specifies that each node in sequence in the first node set is to be replaced by the associated node in the second set. This operation may be used to join separate portions of a finite element model.

Elements may be excluded from failure using the following general statement

***EXCLUDE ELEMENT**
ELSET1, ELSET2, ... , ELSETn

where **ELSETi** are element set identifiers. Any number of element sets may be entered on each line. The total number of input sets is currently limited to 100. This command specifies that all elements specified in the various sets be excluded from failure. This feature may be used for models where only failure in a portion of the model is of interest, or regions of the overall model may be coarsely modeled and local failure in these regions is not to be accounted for in the prediction of residual strength.

To terminate parsing of the input file, the following statement is required:

***ENDDATA**

Any statements following ***ENDDATA** in the input file are ignored.

2.4 Execution Modification

Various modifications to the execution of solution procedures may be specified through the use of the ***PARAMETER** statement.

2.4.1 User-Defined Buckling Mode Interpretation

RESTRAN has built-in algorithms for processing and interpreting sublaminate buckling modes. These algorithms, however, assume that delaminations form a planar surface. As described in the RESTRAN users manual [3], this surface need not be continuous, but a delamination which cannot be described by a sequence of plane sections cannot be processed. To overcome this limitation, a user-written subroutine may be used to analyze the buckling mode of complicated delamination configurations. Specialized interpretation of simultaneous buckling of multiple delaminations or stepped fracture surfaces, such as depicted in Figure 23, may then be performed.

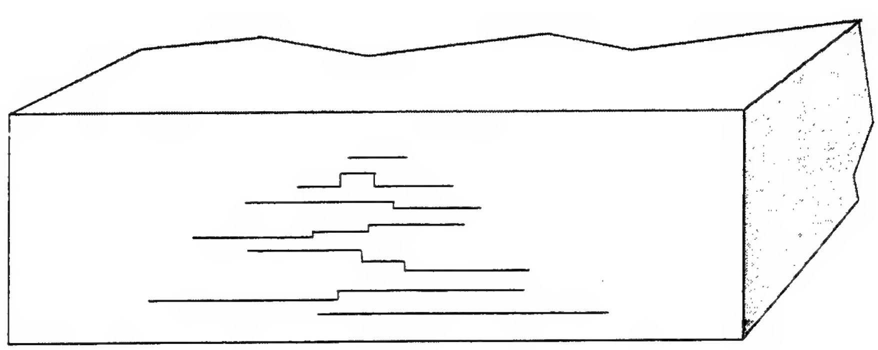


Figure 23. Complex delamination states.

To invoke the access of a user-written subroutine to analyze buckling modes, the following entry is included anywhere in the input stream.

***PARAMETER USRBKL**

This causes RESTRAN to bypass the existing interpretive algorithms and access two user-defined routines. These subroutine are designated **USERB1** and **USERB2**. The routine **USERB1** is accessed to analyze the current buckling mode shape and determine whether the buckling mode is acceptable and should be used as a possible instability failure mode. The format of this routine is given in Figure 24.

```
      SUBROUTINE USERB1 ( EMODE,NELEM,NDSET,NXSET,NDID,EIGV,NUMNOD,
     1                    NBPRC,LCNSL,ITERB,NUMEXC,NACC,NSURF,NINC,
     2                    MDXO,MDX1,MAXB,MDX3 )
C
C      ************************************************
C      **                                            **
C      **    USER-DEFINED SUBROUTINE TO EVALUATE     **
C      **    BUCKLING FAILURE MODE                   **
C      **                                            **
C      ************************************************
C
      IMPLICIT DOUBLE PRECISION(A-H,O-Z)
C
      DIMENSION EMODE(MDXO,MAXB),NELEM(MDX3,12),NDSET(MDX1)
      DIMENSION NXSET(MDX1)
C
C    *** SOURCE CODE FOR EVALUATING BUCKLING FAILURE ***
C
      RETURN
      END
```

Figure 24. **USERB1** subroutine format.

This routine is passed the mode shape contained in the emode array stored as

NODE	POSITIVE NORMAL			NODE	NEGATIVE NORMAL		
ID	U	V	W	ID	U	V	W
N_1	U_1	V_1	W_1	M_1	U_1	V_1	W_1
N_2	U_2	V_2	W_2	M_2	U_2	V_2	W_2
N_3	U_3	V_3	W_3	M_3	U_3	V_3	W_3
$\vdots$	$\vdots$	$\vdots$	$\vdots$	$\vdots$	$\vdots$	$\vdots$	$\vdots$
N_n	U_n	V_n	W_n	M_n	U_n	V_n	W_n

where the displacements are stored in each record as coincident node pairs, N_i & M_i, corresponding to the positive and negative normal to the delamination surface. NDSET is a linear array containing node numbers defining the current delamination, NXSET is an array to store NUMEXC nodes in the delamination set at which coincident nodes are to be removed to satisfy contact constraints. NDID is the node set ID, EIGV is the eigenvalue of the buckling mode, NODNUM is the total number of nodes in the model, NBPRC and ITERB store the buckling mode interpretation procedure flag and number of iterations, and LCNSL is the current iteration count for the delamination. NACC is set equal to zero if the mode shape is unacceptable and equal to one if this buckling mode is to be considered in determining the next failure mode. NSURF is a parameter that will be passed into the user-written subroutine USERB2 which will assess element failures in the buckled sublaminate. The format of the USERB2 user-defined subroutine is given in Figure 25.

```
      SUBROUTINE USERB2 ( NELEM,NODSET,NSETI,NDELM,NDID,NELFL,
     1                    ELFAC,NFAIL,NUMELF,NSURF,MAX1,MAX2,
     2                    MAX3,MAX4 )
C
C     ************************************************
C     **                                            **
C     **     USER DEFINED SUBROUTINE TO DETERMINE   **
C     **     ELEMENT FAILURE DUE TO BUCKLING        **
C     **                                            **
C     ************************************************
C
      IMPLICIT DOUBLE PRECISION (A-H,O-Z)
C
      DIMENSION NELEM(MAX3,12),NODSET(MAX2)
      DIMENSION NELFL(MAX1),ELFAC(MAX1),NDELM(MAX4)
C
      *** USER DEFINED CODE TO SET ELEMENT ***
      *** FAILURE DUE TO BUCKLING          ***
C
      RETURN
      END
```

Figure 25. **USERB2** subroutine format.

The array NELEM contains the element connectivity data. The format of this array is

NELID	N1	N2	N3	N4	N5	N6	N7	N8

in which NELID is the element number, and N1$\rightarrow$ N8 are the node numbers. These node numbers have been renumbered internally. To obtain the input node numbers, the conversion table, NSETI, can be used. The original input node number for the j^{th} node and the i^{th} element is obtained as

MPDE = MSETI(NELEM(I,J+1))

The nodset array contains the node ID's defining the current delamination. This array has the form:

NODSET(NSID, N1, N2, ... ,Nn)

where NSID is the node set ID, and Ni are the node numbers. The routine assigns element failure such that NELFM(I) stores the i^{th} element ID, and the property reduction factor for this element is stored in ELFAC(I). NDID is the delamination ID number. NUMELF is the total number of elements that have been determined to fail, which is set by the user subroutine.

Examples of user-written subroutines to interpret buckling modes and assign element failures are presented in Appendices D and E.

2.4.2 Element Failure Exclusion

***PARAMETER LAPFAIL**

This parameter causes the elements to which external loads are applied to not be analyzed for failure. If an element is completely failed to which external forces are applied, the definition of residual strength used in RESTRAN involving a scalar multiplier to the initial load vector loses definition and the analysis is terminated. Although these particular elements may be identified and assigned moduli and/or strength ultimates to preclude failure, in certain cases where elements in this set are involved in the final catastrophic failure event, RESTRAN will use the material constants associated with these elements and predict an erroneously large strength measure before terminating. Thus, the use of this parameter is generally recommended but is maintained as an option.

2.4.3 Job Execution Status

In running an iterative solution sequence which may involve lengthy execution times for large models, the following entry can be included to request an intermediate record of where execution is proceeding

***PARAMETER STATUS**

The additional output containing execution description and timing information is written to the file *restran.log*.

2.5 Output Control

Depending on the operating system used to run RESTRAN, partitions set up for scratch files may be inadequate to hold the temporary files generated during solution. A parameter may be input to direct the directory in which scratch files are to be stored. This parameter is given by

***PARAMETER DIRECTORY = FPATH**

where **FPATH** may be set equal to CURRENT or SCRATCH. If the CURRENT option is used, all temporary files are created in the current directory in the **fort.*** format and must be manually deleted. The default is SCRATCH.

Nodal displacement output is specified using the following statement:

***NODE PRINT, NODSET = NDS**
NSTRING

where **NSTRING** is a character string containing the characters **D** and/or **M** in any order to print displacements and/or buckling mode shapes, respectively, after convergence of the i^{th} load increment. The optional parameter **NDS** may be used to specify a node set number to limit output.

Output of element stress and strain data is requested using the statement

***ELEMENT PRINT, OUTPUT=LOC, SYSTEM=CRD, ELSET=NES**
ESTRING

where **ESTRING** is a character string containing the characters **S** and/or **E** in any order to print stresses and/or strains, respectively. The optional character parameter **LOC** may be set to 'GAUSS' (default) or 'LAYER' for output at element Gauss points or at the midpoint of ply layers, respectively. The character parameter **CRD** may be set to 'LOCAL' for output in the local element coordinate system or 'GLOBAL' (default) for output in the global coordinate system. The integer parameter **NES** may be used to specify an element set to limit output.

In an iterative residual strength analysis, ply failure statistics can be output after convergence at each analysis cycle through the following statement:

***PLY FAILURE PRINT, NELSET=NPS**

where ply failures in each element are output showing the percentages of failures due to matrix, fiber, general, and buckling failure. The optional parameter **NPS** may be used to specify an element set number to limit output. This statement may also be used in a linear static analysis to cause the computation of element failure indices after stress recovery is completed. Output is limited to the element ply exhibiting the maximum failure index, the predicted failure mode, and the associated factor to scale the applied loads to cause failure of the ply. At the end of the list, the minimum scale factor is identified and may be considered a residual strength measure based on an overall first-ply failure criterion.

2.6 Graphics

The graphics option in RESTRAN has been developed with two primary aims. First, because of the specialized nature of RESTRAN in predicting progressive failure, the graphical information of interest is assumed to concentrate on the element level failure depictions together with overall deformation and/or local buckling mode shapes. This consideration limited the need to provide detailed graphical information on local stress/strain/deformation contours as is available using commercial pre- and postprocessing programs such as PATRAN, which are designed for general structural analysis. Secondly, a commercial software package in widespread use with graphical capabilities and available for use on computer platforms down to the PC level was also of interest. This would free the essential operations of the RESTRAN analysis program from being linked to built-in, platform-dependent graphics routines which would require several versions of the program to be released. Therefore, graphics output can be selected in two formats: MATHEMATICA and TECPLOT. Both are common commercial codes which offer a robust capability in processing primitive graphical objects such as polygonal surfaces and wireframe meshes. The generated output file can be directly input into the selected external code for viewing. The MATHEMATICA format offers more parameters for viewing the input structural model and progressive failure through a color coding and gray-scale scheme. This input file contains complete information on the structural geometry and can be edited to change any viewing parameters such as those effecting view orientation, lighting, and magnification factor for deformation/mode shape depiction. The graphics option is invoked by including the following statement in the input file:

***GRAPHICS, FORMAT = FMT**
NDFRM, NWTRN, GMFAC

where the parameter, **FMT**, is set equal to '**MATHEMATICA**', '**TECPLOT**', or '**USER-DEFINED**' to specify the required format of the output file. Additional optional parameters are given by

NDFRM = 1 For only undeformed geometry
= 2 For both deformed and undeformed geometry
NWTRN = 1 For a wireframe depiction in PREPASS
= 2 For surface depiction in PREPASS
GMFAC = Magnification factor for deformed plots

In Prepass, RESTRAN will generate a solid or wireframe depiction of the model, and output is written to a file designated *restran.prp*. In any other solution option, RESTRAN will generate deformation/mode shape graphics for linear static or instability analysis. For residual strength analysis, a visual depiction of damage state and deformation plot will be generated at the end of each converged progressive failure cycle. This information is stored in a file designated *restran.grf*. These files may then be directly input into the selected graphics program to generate the graphical display. An example of a generated graphics file in MATHEMATICA format is presented in Appendix F.

The "USER-DEFINED" option causes RESTRAN to call the user-written subroutine USRGRF. The format of this subroutine is shown in Figure 26.

```
      SUBROUTINE USRGRF (NELEM,COORDS,U,NSETI,NELNUM,NUMNOD,
     1                    NWTRN,NDFRM,GMFAC,SCALE,NPATH,NTG,
     2                    NFLAG,NDEV,MDX1,MDX2,MDX3 )
C
C     ******************************************
C     **                                      **
C     **  USER-DEFINED GRAPHICS OUTPUT ROUTINE  **
C     **                                      **
C     ******************************************
C
      IMPLICIT DOUBLE PRECISION(A-H,O-Z)
C
      DIMENSION NELEM(MDX3,12),COORDS(MDX2,4),U(MDX1)
      DIMENSION NSETI(MDX1)
C
C
C     NODE CONVENTION
C
C                           Y,V
C             Z,W          /
C             |           /              SURFACE
C             |   8---------7        FACE   NODES
C             |  /|        /|         F1: (1,2,6,5)
C             | / |       / |         F2: (2,3,7,6)
C             |/  |      /  |         F3: (3,4,8,7)
C             5---------6   |         F4: (1,5,8,4)
C             |   |     |   |         F5: (1,4,3,2)
C             |   4-----|---3         F6: (5,6,7,8)
C             |  /      |  /
C             | /       | /
C             |/        |/
C             1---------2 --------- X,U
C
C
C      <<< PROGRAM STATEMENTS TO OUTPUT GRAPHICAL >>>
C      <<< INFORMATION IN A USER-DEFINED FORMAT   >>>
C
      RETURN
      END
```

Figure 26. Format of user-defined subroutine **USRGRF**.

In USRGRF, the storage format of element connectivity, coordinate data, and node number conversion contained in the NELEM, COORDS, and NSETI arrays have been previously defined in Section 2.4.1. Other quantities passed into the user-defined routine are the U array which contains

the displacement or mode shape vector and a sequence of scalar parameters. NELNUM and NODNUM contain the number of elements and nodes, respectively. SCALE and NTG store the current load multiplier and global analysis cycle number. NPATH stores the solution procedure currently being processed. This is given by

$$NPATH = \begin{cases} 1 & MaterialFailureCalculation \\ 2 & BucklingFailureCalculation \\ 4 & LinearStaticAnalysis \\ 5 & LinearBucklingAnalysis \end{cases} \tag{18}$$

NFLAG = 1 or 2, depending on whether the solution is in a PREPASS or ANALYSIS mode, respectively. The remaining parameters, MDX1, MDX2, and MDX3 are size parameters set internally by RESTRAN and must not be altered. An example of a **USRGRF** subroutine is presented in Appendix G.

2.7 Miscellaneous

To include a heading or title to the program output, the following input record is used:

***HEADING**
JOB TITLE

where the job title may be up to 80 characters in length.

For convenience, the input file may be included with the program output by including the following statement

***ECHO**

This will echo the input file verbatim to the output stream.

Comments may be inserted between statements anywhere in the input file using a '**' format in fields 1 and 2. For example

** This line is a comment

will cause the input file interpreter to skip this line. Comments may also be inserted within the range of input statements that have a fixed number of entries per record and can have an arbitrary number of records or lines. These include the ***NODE**, ***ELEMENT**, and ***BOUNDARY** statements.

Model data created in programs such as PATRAN in ABAQUS format may be directly included by using the following statement

***INCLUDE, FILE='filename.inp'**

This will cause input to be read from the specified file. The filename must be enclosed in quotes. The included file may contain any input statements that are equivalent to ABAQUS format such as ***NODE**, ***ELEMENT**, ***BOUNDARY**, or ***CLOAD**. All optional parameter inputs should

be in accordance with RESTRAN input formats and, in addition, the *ENDDATA or *END STEP line must be present in the included file to return IO processing to the standard RESTRAN input file *restran.inp*.

3 RESTRAN Error Checking

An extensive set of internal testing of initial input and run-time operations has been incorporated into RESTRAN to alert the user to inconsistencies and errors encountered in the model generation and execution of the various RESTRAN analysis options. The range of error testing is described in the following subsections.

3.1 Tests on Initial Input

The essential tests on the basic input involve checking whether the RESTRAN size parameters are adequate to process the current model and verify that a valid model has been input. Testing of model validity includes checking node definition, element connectivity, assigned material properties, support conditions, and failure criteria. These tests are designed to ensure that a viable model can be created for performing subsequent analyses. Fatal errors are determined as those that preclude the subsequent operation of an analysis algorithm and will cause program execution to cease. Warnings are issued if inconsistencies are detected, such as unused nodes, but execution is allowed to proceed.

3.2 Run-Time Error Determination

A valid, consistent structural model may still be ill-posed and produce meaningless results. During execution, numerous tests are made to assess elastic and differential stiffness properties, eigenmode determination and interpretation, material degradation, and processing performed by user-defined subroutines. Although any set of error checking is bound to be incomplete, the internal testing in RESTRAN will alert the user to the most simple and to many subtle errors in the analysis of residual strength.

4 Input File Format

The RESTRAN input file is designated *restran.inp*. The file format has been implemented with a degree of flexibility in the parsing of input such that all statements are case insensitive, and arbitrary space or comma delimitators can be used in listing statement options and data items. The input stream in RESTRAN is parsed sequentially and may generally be input in any order. The only exceptions are the node and element generation statements ***NGEN**, ***NFILL**, ***ELGEN**, and ***MODEL GENERATION**, which are invoked during the input parsing. The routines that are invoked require node or element data to have been read prior to performing the requested node or element generation. Termination of input is specified by the ***ENDDATA** entry.

5 Computer Implementation of RESTRAN

RESTRAN is written in FORTRAN 77 with the aim of providing maximum portability across different computer platforms. There is no overriding driver or script to automate various support

or implementation functions, such as setting array size parameters and processing user-defined subroutines. The installation of RESTRAN requires that a master memory size parameter be set in the preface to the MAIN section of the RESTRAN code. This parameter is denoted MAXRAM and represents the maximum number of double precision words that can be held in core memory. This segment of code is shown in Figure 27.

```
C   ############################################################
C   @@@                                                      @@@
C   @@@  RESTRAN, REsidual STRength ANalysis, IS A FINITE    @@@
C   @@@  ELEMENT BASED PROGRAM TO PREDICT THE RESIDUAL       @@@
C   @@@  STRENGTH OF IMPACT DAMAGED COMPOSITE LAMINATES      @@@
C   @@@                                                      @@@
C   ############################################################
C
      IMPLICIT DOUBLE PRECISION(A-H,O-Z)
C
C          ##############################################
C          @@@                                        @@@
C          @@@       RESTRAN MEMORY PARMETERS         @@@
C          @@@                                        @@@
C          ##############################################
C
C <<<          SIZE PARAMETERS USED IN PROGRAM:        >>>
C <<<                                                  >>>
C <<< MAXRAM:  MAXIMUM AVAILABLE RAM STORAGE IN        >>>
C <<<          DOUBLE PRECISION WORDS                  >>>
C
      PARAMETER ( MAXRAM = 16500000 )
```

Figure 27. MAXRAM parameter in RESTRAN.

During program execution, judicious segments of RAM memory are internally allocated to perform the preface operations involving model input and processing. Minimum array sizes are automatically determined, and memory is reallocated to minimize both internal storage requirements and the size of external disk files to streamline the execution of a RESTRAN analysis.

User-defined subroutines must be created using a standardized subroutine parameter interface. For convenience, sample routines are provided in separate appendices. These routines need not be written in FORTRAN, but if other languages are used, care must be given to differences in conventions such as how arrays are ordered as in C. These routines must then be compiled and linked with the RESTRAN object code to form an executable module.

6 Computational Efficiency

The computational efficiency of RESTRAN is primarily dictated by the matrix operations associated with decomposition and eigenvalue extraction. As discussed in more detail in the RESTRAN Theoretical Manual [3], the finite element basis naturally leads to the generation of large, sparse,

banded matrices in representing the complete model. Internal algorithms have been created to process matrix storage modes in various formats. These include full matrix storage for small problems, half-bandwidth storage format (default), and out-of-core storage for large problems that exceed the internal RAM memory capacity and require most data storage to be resident in external disk files. The basic determinator of execution speed is the selection of internal data storage format required to hold sufficient data in high-speed in-core memory. Bandwidth storage assumes enough RAM memory to store a complete global matrix at any one time with intermittent I/O operations to swap one global matrix quantity for another. The out-of-core solution mode is slowest due to the high I/O overhead of transferring data between core and external files which must be performed constantly during program execution.

Additional considerations pertain to the direct assembly procedure currently used, which is more computationally and I/O costly than frontal solution methods, and to the condensation of internal equations involved in isolating individual delaminations and imposition of applied displacement boundary conditions. These constraints are applied using row and column shifting, which is more intensive than simple equation isolation using unit diagonal-zeroed row/column procedures that are straightforward in linear equation solution but introduce difficulties in performing eigenanalyses.

Thus, the execution time of performing an analysis using RESTRAN is significantly higher than that obtained in performing similar solution procedures in the widely used commercial codes such as NASTRAN, ABAQUS, and ANSYS, and may be addressed in future enhancements. The specialized analysis of residual strength accounting for combined progressive material and structural failure performed by the RESTRAN code is, however, simply not available in a single commercial program.

7 Numerical Examples

The following subsections present sample problems illustrating the use of the RESTRAN code. Each problem uses a model of a simple laminated composite casing in order to contrast different finite element discretizations, applied loading, existing damage, and analysis procedures. Timing summaries are provided to demonstrate the performance of RESTRAN in a Linux environment on a Carrera workstation with a 530 MHz Alpha cpu.

7.1 Linear Static Analysis

A static analysis of the finite element model shown in Figure 25 is performed. An arbitrary layup based on a nominal ply thickness of 0.0052in was selected for the various model sections. The top and bottom plates were composed of $(90_6/\pm45_4 \mid \pm45/0_{28} \mid)_s$ laminates where '|' designates element layer interfaces. The vertical panel laminate was modeled as $(0_6/\pm45_4/90_5/\pm45_5 \mid 0_6/\pm45_5/0_4 \mid)_s$. Ply properties were selected as

$$\begin{array}{lcllcllcl} E_1 & = & 2.48E6 & E_2 & = & 2.48E6 & E_3 & = & 0.71E6 \\ G_{23} & = & 3.0E5 & G_{13} & = & 3.0E5 & G_{12} & = & 9.9E5 \\ \nu_{23} & = & 0.28 & \nu_{13} & = & 0.28 & \nu_{12} & = & 0.243 \end{array}$$

For first ply failure, the maximum stress criterion was selected with the following material strengths

$$\begin{array}{lllllllll} X_t & = & 21.3E3 & Y_t & = & 21.3E3 & Z_t & = & 21.2E3 \\ X_c & = & 47.8E3 & Y_c & = & 47.8E3 & Z_c & = & 21.0E3 \\ R & = & 1.65E3 & S & = & 1.65E3 & T & = & 7.0E3 \end{array}$$

The model input, job log, and abbreviated output are shown subsequently, together with the resulting deformed plot.

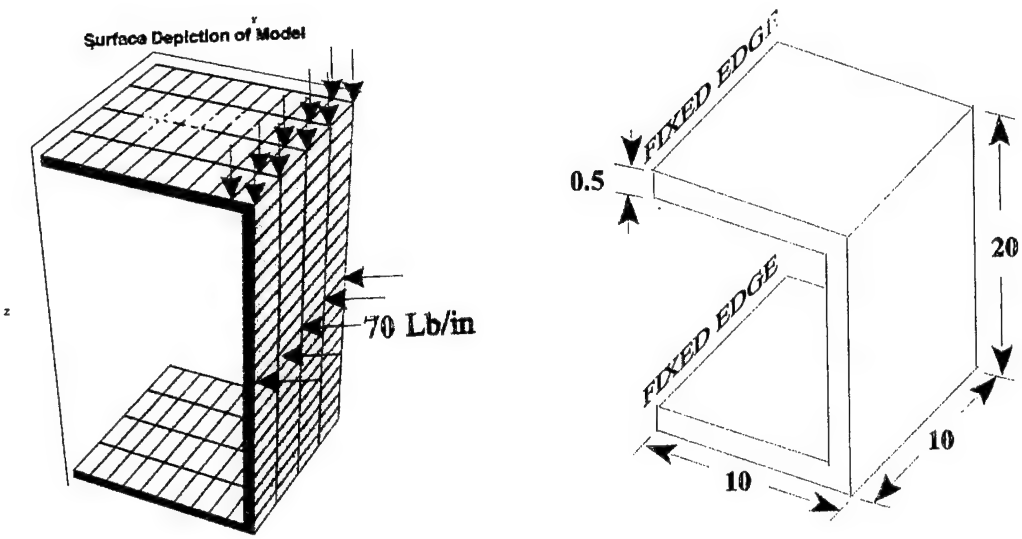


Figure 28. Model used for linear static analysis.

7.1.1 Input Data File for a Linear Static Problem

```
*HEADING
 MODEL OF A COMPOSITE CASING
*SOLUTION, METH = LSA
 6,15
**ECHO
**PREPASS
*NODE PRINT
  D,M
*PLY FAILURE PRINT
*MEMORY ALLOCATION
 BAND
*GRAPHICS, format = mathematica
 2, 2, 1.0
**
**
**   NODE DEFINITIONS
**
*node
 1      0.0      0.0    20.0
 5      0.0     10.0    20.0
```

```
 51     10.0     0.0    20.0
 55     10.0    10.0    20.0
 151    10.0     0.0     0.0
 155    10.0    10.0     0.0
 201     0.0     0.0     0.0
 205     0.0    10.0     0.0
 1001    0.0     0.0    19.875
 1005    0.0    10.0    19.875
 1051    9.875   0.0    19.875
 1055    9.875  10.0    19.875
 1151    9.875   0.0     0.125
 1155    9.875  10.0     0.125
 1201    0.0     0.0     0.125
 1205    0.0    10.0     0.125
 2001    0.0     0.0    19.75
 2005    0.0    10.0    19.75
 2051    9.75    0.0    19.75
 2055    9.75   10.0    19.75
 2151    9.75    0.0     0.25
 2155    9.75   10.0     0.25
 2201    0.0     0.0     0.25
 2205    0.0    10.0     0.25
 3001    0.0     0.0    19.625
 3005    0.0    10.0    19.625
 3051    9.625   0.0    19.625
 3055    9.625  10.0    19.625
 3151    9.625   0.0     0.375
 3155    9.625  10.0     0.375
 3201    0.0     0.0     0.375
 3205    0.0    10.0     0.375
 4001    0.0     0.0    19.5
 4005    0.0    10.0    19.5
 4051    9.5     0.0    19.5
 4055    9.5    10.0    19.5
 4151    9.5     0.0     0.5
 4155    9.5    10.0     0.5
 4201    0.0     0.0     0.5
 4205    0.0    10.0     0.5
**
**   GENERATE NODE SETS
**
*NGEN, NID=10
1, 5, 1
*NGEN, NID=20
51, 55, 1
*NGEN, NID=30
151, 155, 1
*NGEN, NID=40
201, 205, 1
*NGEN, NID=50
1001, 1005, 1
*NGEN, NID=60
1051, 1055, 1
*NGEN, NID=70
```

```
1151, 1155, 1
*NGEN, NID=80
1201, 1205, 1
*NGEN, NID=90
2001, 2005, 1
*NGEN, NID=100
2051, 2055, 1
*NGEN, NID=110
2151, 2155, 1
*NGEN, NID=120
2201, 2205, 1
*NGEN, NID=130
3001, 3005, 1
*NGEN, NID=140
3051, 3055, 1
*NGEN, NID=150
3151, 3155, 1
*NGEN, NID=160
3201, 3205, 1
*NGEN, NID=170
4001, 4005, 1
*NGEN, NID=180
4051, 4055, 1
*NGEN, NID=190
4151, 4155, 1
*NGEN, NID=200
4201, 4205, 1
**
** GENERATE NODE PLANES
**
*NFILL
10,  20,   9, 5
20,  30,  19, 5
30,  40,   9, 5
50,  60,   9, 5
60,  70,  19, 5
70,  80,   9, 5
90,  100,  9, 5
100, 110, 19, 5
110, 120,  9, 5
130, 140,  9, 5
140, 150, 19, 5
150, 160,  9, 5
170, 180,  9, 5
180, 190, 19, 5
190, 200,  9, 5
**
**        DEFINE GENERATOR ELEMENTS
**
** top horizontal component
**
*ELEMENT,  LAYUP = 1
   1, 1001, 1006, 1007, 1002,    1,     6,     7,     2
*ELEMENT,  LAYUP = 2
```

```
 100, 2001, 2006, 2007, 2002, 1001, 1006, 1007, 1002
*ELEMENT,  LAYUP = 3
 200, 3001, 3006, 3007, 3002, 2001, 2006, 2007, 2002
*ELEMENT,  LAYUP = 4
 300, 4001, 4006, 4007, 4002, 3001, 3006, 3007, 3002
**
** bottom horizontal component
**
*ELEMENT,  LAYUP = 1
 400,  201,  196,  197,  202, 1201, 1196, 1197, 1202
*ELEMENT,  LAYUP = 2
 500, 1201, 1196, 1197, 1202, 2201, 2196, 2197, 2202
*ELEMENT,  LAYUP = 3
 600, 2201, 2196, 2197, 2202, 3201, 3196, 3197, 3202
*ELEMENT,  LAYUP = 4
 700, 3201, 3196, 3197, 3202, 4201, 4196, 4197, 4202
**
** vertical component
**
*ELEMENT, LAYUP = 5, orientation = 1
 800, 1151, 151,  152,  1152, 1146, 146,  147,  1147
*ELEMENT, LAYUP = 6, orientation = 1
 900, 2151, 1151, 1152, 2152, 2146, 1146, 1147, 2147
*ELEMENT, LAYUP = 7, orientation = 1
1000, 3151, 2151, 2152, 3152, 3146, 2146, 2147, 3147
*ELEMENT, LAYUP = 8, orientation = 1
1100, 4151, 3151, 3152, 4152, 4146, 3146, 3147, 4147
**
**   ELEMENT GENERATION
**
*ELGEN, ELID = 10
  1,10,5,1,4,1,10,,,
100,10,5,1,4,1,10,,,
200,10,5,1,4,1,10,,,
300,10,5,1,4,1,10,,,
*ELGEN, ELID = 20
 800,0,0,0,4,1,1,20,-5,4
 900,0,0,0,4,1,1,20,-5,4
1000,0,0,0,4,1,1,20,-5,4
1100,0,0,0,4,1,1,20,-5,4
*ELGEN, ELID = 30
400,10,-5,5,4,1,4,,,
500,10,-5,5,4,1,4,,,
600,10,-5,5,4,1,4,,,
700,10,-5,5,4,1,4,,,
**
** DEFINE SET OF ELEMENTS AT UPPER AND LOWER PLATE JUNCTIONS
**
*ELSET, ESID = 500
10   20   30   40   109  119  129  139  209  219  229  239
309  319  329  339  445  449  453  457  545  549  553  557
645  649  653  657  745  749  753  757  800  801  802  803
900  901  902  903  1000 1001 1002 1003 1100 1101 1102 1103
876  877  878  879  976  977  978  979  1076 1077 1078 1079
```

```
1176 1177 1178 1179
**
** EXCLUDE ELEMENT SET 500 FROM FIRST PLY FAILURE PREDICTION
**
*EXCLUDE ELEMENTS
 500
**
** RELAX TOLERANCE IN DEFORMED GEOMETRY CHECKS
**
*DEFORMED GEOMETRY
20.0
**
** LAMINATE DESCRIPTION
**
*LAYER, LAYUP = 1
 1, 0.0416,  90.0
 1, 0.0208,  45.0
 1, 0.0208, -45.0
*LAYER, LAYUP = 2
 1, 0.0208,  45.0
 1, 0.0208, -45.0
 1, 0.1456,   0.0
*LAYER, LAYUP = 3
 1, 0.1456,   0.0
 1, 0.0208, -45.0
 1, 0.0208,  45.0
*LAYER, LAYUP = 4
 1, 0.0208, -45.0
 1, 0.0208,  45.0
 1, 0.0416,  90.0
**
*LAYER, LAYUP = 5
 1, 0.0312,   0.0
 1, 0.0208,  45.0
 1, 0.0208, -45.0
 1, 0.0260,  90.0
 1, 0.0260,  45.0
 1, 0.0260, -45.0
*LAYER, LAYUP = 6
 1, 0.0312,   0.0
 1, 0.0260,  45.0
 1, 0.0260, -45.0
 1, 0.0208,   0.0
*LAYER, LAYUP = 7
 1, 0.0208,   0.0
 1, 0.0260, -45.0
 1, 0.0260,  45.0
 1, 0.0312,   0.0
*LAYER, LAYUP = 8
 1, 0.0260, -45.0
 1, 0.0260,  45.0
 1, 0.0260,  90.0
 1, 0.0208, -45.0
 1, 0.0208,  45.0
```

```
 1, 0.0312,   0.0
**
**   MATERIAL DEFINITION
**
*MATERIAL, MATID = 1
  2.48E6,2.48E6,0.71E6,0.3E6,0.3E6,0.99E6
  0.28,0.28,0.243
*FAILURE CRITERIA, FCID = 1
 MAX-STRESS
 21.2E3, 47.8E3, 21.2E3, 47.8E3, 21.2E3, 21.0E3
 1.65E3, 1.65E3, 7.0E3
*DAMAGE LAW, DLID = 1
 NULL
**
**  ESTABLISH LOCAL COORDINATE SYSTEM FOR VERTICAL SECTION
**
*ORIENTATION
 1,  0.0, 1.0, 0.0,  0.0, 0.0, 1.0
**
*BOUNDARY2
  10, 1, 1
  50, 1, 1
  90, 1, 1
 130, 1, 1
 170, 1, 1
*BOUNDARY2
  40, 1, 3
  80, 1, 3
 120, 1, 3
 160, 1, 3
 200, 1, 3
**
*CLOAD
 46, 3,  -50.0
 47, 3, -100.0
 48, 3, -100.0
 49, 3, -100.0
 50, 3,  -50.0
 51, 3,  -50.0
 52, 3, -100.0
 53, 3, -100.0
 54, 3, -100.0
 55, 3,  -50.0
101, 1, -100.0
102, 1, -200.0
103, 1, -200.0
104, 1, -200.0
105, 1, -100.0
**
*ENDDATA
```

7.1.2 Timing Summary of Job Execution

```
@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@
@@                                           @@
@@        U.S. ARMY RESEARCH LABORATORY      @@
@@                                           @@
@@              R E S T R A N                @@
@@                                           @@
@@   RESIDUAL STRENGTH ANALYSIS OF IMPACT    @@
@@       DAMAGED COMPOSITE LAMINATES         @@
@@                                           @@
@@               VERSION 1.0                 @@
@@                                           @@
@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@


   M O D E L   S I Z E   P A R A M E T E R S

      NUMBER OF ELEMENTS =        640
      NUMBER OF NODES    =       1025
      DEGREES OF FREEDOM =       3075
      SYSTEM BANDWIDTH   =        183


TOTAL RAM MEMORY AVAILABLE =   14500000 WORDS

TOTAL RAM MEMORY REQUIRED  =     784080 WORDS

*****************************************
**                                     **
** TIMING SUMMARY OF PROGRAM EXECUTION **
**                                     **
*****************************************

RESTRAN PREFACE OPERATIONS:   5.3739 SECONDS

@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@
@@@                                   @@@
@@@ BEGINNING LINEAR STATIC ANALYSIS @@@
@@@                                   @@@
@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@


ELASTIC STIFFNESS ASSEMBLY TIME:       13.8543 SECONDS

BOUNDARY CONSTRAINT PROCESSING TIME:    0.8140 SECONDS

LINEAR SOLUTION TIME:                   2.1745 SECONDS

STRESS RECOVERY TIME:                  13.9383 SECONDS

GRAPHICAL OUTPUT GENERATION TIME:       2.6908 SECONDS
```

TOTAL JOB EXECUTION TIME: 43.2183 SECONDS

7.1.3 Linear Static Solution Output File

```
@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@
@@                                          @@
@@      U.S. ARMY RESEARCH LABORATORY       @@
@@                                          @@
@@              R E S T R A N               @@
@@                                          @@
@@   RESIDUAL STRENGTH ANALYSIS OF IMPACT   @@
@@       DAMAGED COMPOSITE LAMINATES        @@
@@                                          @@
@@               VERSION 1.0                @@
@@                                          @@
@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@

*** WARNING: MATERIAL ID      1 HAS A MIXED MODE FAILURE CRITERIA
             ASSOCIATED WITH A SINGLE MODE DAMAGE LAW (NULL).
             NULL ACCEPTED

                ######################################
                ###                                ###
                ### M O D E L   D E F I N I T I O N ###
                ###                                ###
                ######################################

                         J O B   H E A D I N G:
                         ---------------------

 MODEL OF A COMPOSITE CASING


                    S O L U T I O N   C O N T R O L:
                    -------------------------------

 S O L U T I O N   A L G O R I T H M:

   LINEAR STATIC ANALYSIS

 M E M O R Y   A L L O C A T I O N:

      BANDWIDTH STORAGE MODE

     * BANDWIDTH MINIMIZATION WILL BE PERFORMED

                P R O G R A M   O P T I O N S:
                -----------------------------
```

```
* EXTENDED JOB EXECUTION STATISTICS WILL BE PRINTED

* THE FOLLOWING ELEMENT SETS ARE EXCLUDED FROM FAILURE PREDICTION

 500

      R E Q U E S T E D   P R I N T   O P T I O N S:
      ---------------------------------------------


* PRINT DISPLACEMENTS AFTER CONVERGENCE AT THE iTH LOAD INCREMENT

* PRINT BUCKLING MODE SHAPE CORRESPONDING TO THE LOWEST BUCKLING
  LOAD MULTIPLIER AT THE iTH ITERATION

* PRINT ELEMENT PLY FAILURE STATISTICS


                    G R A P H I C S:
                    ----------------

@ GRAPHICAL OUTPUT IN MATHEMATICA FORMAT WILL BE GENERATED

@ BOTH UNDEFORMED AND DEFORMED GEOMETRY IS REQUESTED
  DEFORMATION/MODE MAGNIFICATION FACTOR = 0.1000E+01

@ RESTRAN GRAPHICS CONTAINED IN FILE: restran.grf


            N O D E   D E F I N I T I O N S:
            -------------------------------

   NODE ID        X           Y           Z

        1     0.000E+00   0.000E+00   0.200E+02
        2     0.000E+00   0.250E+01   0.200E+02
        3     0.000E+00   0.500E+01   0.200E+02
        4     0.000E+00   0.750E+01   0.200E+02
        5     0.000E+00   0.100E+02   0.200E+02
                              .
                              .
                              .
     4201     0.000E+00   0.000E+00   0.500E+00
     4202     0.000E+00   0.250E+01   0.500E+00
     4203     0.000E+00   0.500E+01   0.500E+00
     4204     0.000E+00   0.750E+01   0.500E+00
     4205     0.000E+00   0.100E+02   0.500E+00


          S P E C I F I E D   N O D E   S E T S:
          ------------------------------------

                   SET ID    10:
```

1	2	3	4	5

SET ID 20:

51	52	53	54	55

.
.
.

SET ID 190:

4151	4152	4153	4154	4155

SET ID 200:

4201	4202	4203	4204	4205

E L E M E N T D E F I N I T I O N S:

ELEMENT	N1	N2	N3	N4	N5	N6	N7	N8
1	1001	1006	1007	1002	1	6	7	2
2	1006	1011	1012	1007	6	11	12	7
3	1011	1016	1017	1012	11	16	17	12
4	1016	1021	1022	1017	16	21	22	17
5	1021	1026	1027	1022	21	26	27	22
				.				
				.				
				.				
1175	4064	3064	3065	4065	4059	3059	3060	4060
1176	4056	3056	3057	4057	4051	3051	3052	4052
1177	4057	3057	3058	4058	4052	3052	3053	4053
1178	4058	3058	3059	4059	4053	3053	3054	4054
1179	4059	3059	3060	4060	4054	3054	3055	4055

E L E M E N T S E T S:

ELEMENT SET ID 10:

2	3	4	5	...

ELEMENT SET ID 20:

801	802	803	804	...

ELEMENT SET ID 30:

405	410	415	420	...

ELEMENT SET ID 500:

10 20 30 40 ...

M A T E R I A L   P R O P E R T Y   D A T A:

MATERIAL ID 1

E1 = 0.248E+07	G23 = 0.300E+06	MU23 = 0.28000
E2 = 0.248E+07	G13 = 0.300E+06	MU13 = 0.28000
E3 = 0.710E+06	G12 = 0.990E+06	MU12 = 0.24300

ASSOCIATED FAILURE LAW:

MAXIMUM STRESS CRITERION

XT = 0.212E+05	XC = 0.478E+05	YT = 0.212E+05
YC = 0.478E+05	ZT = 0.212E+05	ZC = 0.210E+05
R = 0.165E+04	S = 0.165E+04	T = 0.700E+04

ASSOCIATED DAMAGE LAW:

NULL POST-FAILURE PROPERTIES

M A T E R I A L   L A Y E R   D E F I N I T I O N S

LAYUP ID = 1
NUMBER OF PLIES = 3

MATERIAL ID	PLY THICKNESS	ORIENTATION
1	0.04160	90.00000
1	0.02080	45.00000
1	0.02080	-45.00000

LAYUP ID = 2
NUMBER OF PLIES = 3

MATERIAL ID	PLY THICKNESS	ORIENTATION
1	0.02080	45.00000
1	0.02080	-45.00000
1	0.14560	0.00000

LAYUP ID = 3
NUMBER OF PLIES = 3

MATERIAL ID	PLY THICKNESS	ORIENTATION
1	0.14560	0.00000
1	0.02080	-45.00000

1	0.02080	45.00000

LAYUP ID = 4
NUMBER OF PLIES = 3

MATERIAL ID	PLY THICKNESS	ORIENTATION
1	0.02080	-45.00000
1	0.02080	45.00000
1	0.04160	90.00000

LAYUP ID = 5
NUMBER OF PLIES = 6

MATERIAL ID	PLY THICKNESS	ORIENTATION
1	0.03120	0.00000
1	0.02080	45.00000
1	0.02080	-45.00000
1	0.02600	90.00000
1	0.02600	45.00000
1	0.02600	-45.00000

LAYUP ID = 6
NUMBER OF PLIES = 4

MATERIAL ID	PLY THICKNESS	ORIENTATION
1	0.03120	0.00000
1	0.02600	45.00000
1	0.02600	-45.00000
1	0.02080	0.00000

LAYUP ID = 7
NUMBER OF PLIES = 4

MATERIAL ID	PLY THICKNESS	ORIENTATION
1	0.02080	0.00000
1	0.02600	-45.00000
1	0.02600	45.00000
1	0.03120	0.00000

LAYUP ID = 8
NUMBER OF PLIES = 6

MATERIAL ID	PLY THICKNESS	ORIENTATION
1	0.02600	-45.00000
1	0.02600	45.00000
1	0.02600	90.00000
1	0.02080	-45.00000
1	0.02080	45.00000
1	0.03120	0.00000

M A T E R I A L   C O O R D I N A T E   S Y S T E M S

ID	V11	V12	V13	V21	V22	V23
1	0.000E+00	0.100E+01	0.000E+00	0.000E+00	0.000E+00	0.100E+01

B O U N D A R Y   C O N D I T I O N S:

SINGLE POINT CONSTRAINTS:

NODE	DOF1 TO	DOF2
1	1	1
2	1	1
3	1	1
4	1	1
	.	
	.	
	.	
4202	1	3
4203	1	3
4204	1	3
4205	1	3

APPLIED CONCENTRATED FORCES:

NODE	DIRECTION	MAGNITUDE
46	3	-.500E+02
47	3	-.100E+03
48	3	-.100E+03
49	3	-.100E+03
50	3	-.500E+02
51	3	-.500E+02
52	3	-.100E+03
53	3	-.100E+03
54	3	-.100E+03
55	3	-.500E+02
101	1	-.100E+03
102	1	-.200E+03
103	1	-.200E+03
104	1	-.200E+03
105	1	-.100E+03

FORCE/MOMENT RESULTANTS AT ORIGIN:

|R| = 0.113E+04 |M| = 0.567E+04

Rx = -.800E+03 Ry = 0.000E+00 Rz = -.800E+03

```
Mx = 0.400E+04   My = 0.400E+03   Mz = -.400E+04

Xo = 0.950E+01   Yo = 0.500E+01   Zo = 0.100E+02


*** E N D   O F   M O D E L   D E F I N I T I O N ***

     M O D E L   S I Z E   P A R A M E T E R S

          NUMBER OF ELEMENTS =        640
          NUMBER OF NODES    =       1025
          DEGREES OF FREEDOM =       3075
          SYSTEM BANDWIDTH   =        183


 ###############################################
 ##                                           ##
 ## L I N E A R   S T A T I C   A N A L Y S I S ##
 ##                                           ##
 ###############################################
```

DISPLACEMENT FIELD
(SCALE FACTOR = 0.100E+01)

NODE	U	V	W
1	0.00000E+00	0.27644E-03	-.24278E+00
2	0.00000E+00	0.60853E-04	-.24592E+00
3	0.00000E+00	-.12759E-03	-.24693E+00
4	0.00000E+00	-.31570E-03	-.24594E+00
5	0.00000E+00	-.53044E-03	-.24281E+00
		.	
		.	
		.	
4201	0.00000E+00	0.00000E+00	0.00000E+00
4202	0.00000E+00	0.00000E+00	0.00000E+00
4203	0.00000E+00	0.00000E+00	0.00000E+00
4204	0.00000E+00	0.00000E+00	0.00000E+00
4205	0.00000E+00	0.00000E+00	0.00000E+00

MAXIMUM ELEMENT FAILURE INDICES

ELEMENT	PLY	FAILURE INDEX	MODE	MULTIPLIER
1	3	0.614E-01	GENERAL	0.163E+02
2	3	0.613E-01	GENERAL	0.163E+02
3	3	0.610E-01	GENERAL	0.164E+02
4	3	0.610E-01	GENERAL	0.164E+02
5	3	0.603E-01	GENERAL	0.166E+02
			.	
			.	
			.	
1174	5	0.266E-01	GENERAL	0.377E+02
1175	5	0.322E-01	GENERAL	0.311E+02
1176	1	0.691E-01	GENERAL	0.145E+02

```
   1177       1      0.633E-01    GENERAL  0.158E+02
   1178       1      0.633E-01    GENERAL  0.158E+02
   1179       1      0.692E-01    GENERAL  0.144E+02

HIGHEST GLOBAL FAILURE INDEX OCCURRED IN ELEMENT    902
IN PLY NUMBER    1 PREDICTING FIRST PLY FAILURE AT A
FACTOR TO APPLIED LOADS EQUAL TO 0.174E+01

@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@
@@                                                    @@
@@   S T A T I C   A N A L Y S I S   C O M P L E T E D @@
@@                                                    @@
@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@
```

According to a first ply failure criterion, failure is predicted at a load given by

$$\{P\} = 1.74\{P_{applied}\} \tag{19}$$

A graphical depiction of the deformed shape is shown in Figure 29.

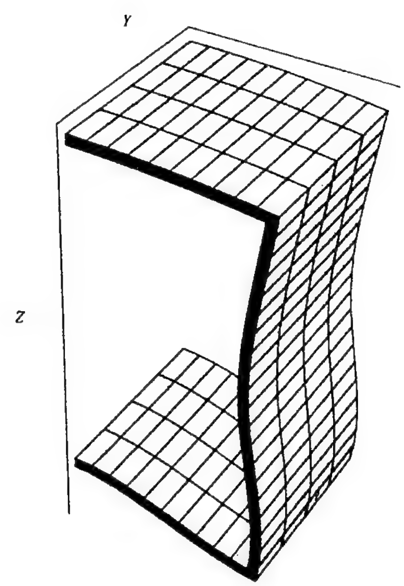


Figure 29. Deformation under applied static loads.

7.2 Linear Buckling Analysis

A similar model of a composite casing as shown in Section 7.1 is analyzed for buckling. The same laminate layup and material properties were used. The model, shown in Figure 30, was constructed to model circular delamination at each of the three interior sublaminate interfaces in the vertical plate. The location of these delaminations are indicated by || in the layup description given by $(0_6/\pm 45_4/90_5/\pm 45_5 \,||\, 0_6/\pm 45_5/0_4 \,||\, 0_4/\mp 45_5/0_6 \,||\, \mp 45_5/90_5/\mp 45_4/0_6$. The instability of these delaminations was analyzed simultaneously. For comparison, the critical buckling load of the entire

component without the presence of delaminations was computed by removing the *DELAMINATION statement from the input file. The model input, job log, and abbreviated output files are shown subsequently, together with the resulting mode shape plots.

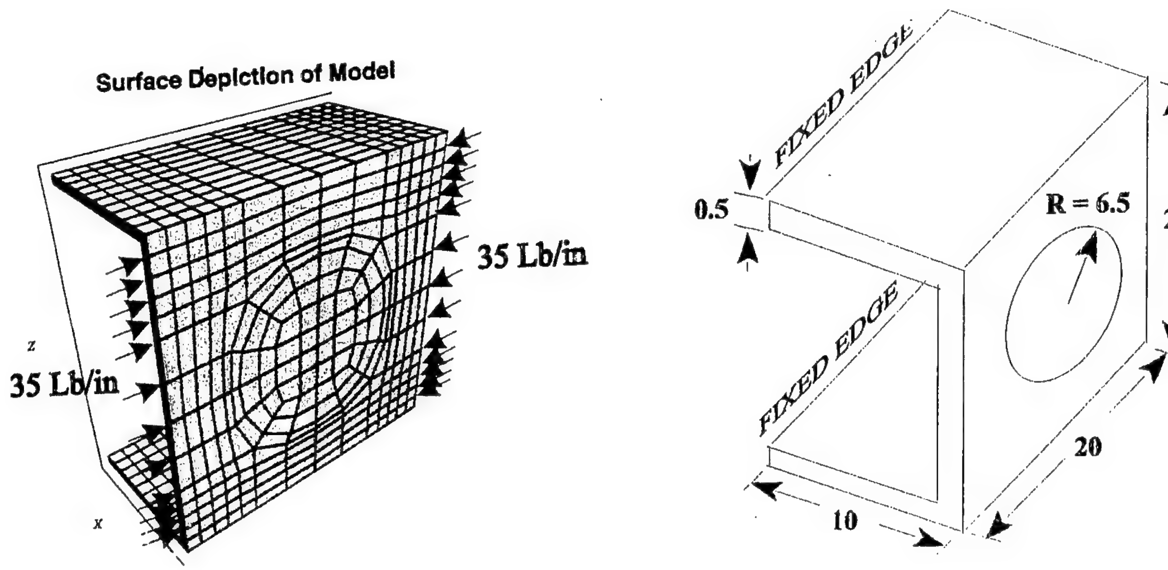


Figure 30. Model used for linear buckling analysis.

7.2.1 Input Data File for a Linear Buckling Problem

```
*HEADING
 STABILITY ANALYSIS OF A DELAMINATED COMPOSITE CASING
*SOLUTION, METH = LBA
 6,15
**ECHO
**PREPASS
*NODE PRINT
  D,M
*MEMORY ALLOCATION
 BAND
*GRAPHICS, format = mathematica
 2, 2, 1.0
**
**   NODE DEFINITIONS
**
*NODE
**
** TOP HORIZONTAL COMPONENT, LEFT
**
   1   0.0    -0.1000005E+02  10.0
   2   0.0    -0.9009584E+01  10.0
   3   0.0    -0.8019119E+01  10.0
```

```
                  .
                  .
**
** TOP HORIZONTAL COMPONENT, RIGHT
**
 151  10.0     -0.1000005E+02  10.0
 152  10.0     -0.9009584E+01  10.0
 153  10.0     -0.8019119E+01  10.0
                  .
                  .
                  .

** BOTTOM HORIZONTAL COMPONENT, LEFT
**
1501   0.0   -0.1000005E+02  -10.0
1502   0.0   -0.9009584E+01  -10.0
1503   0.0   -0.8019119E+01  -10.0
                 .
                 .
                 .
**
** BOTTOM HORIZONTAL COMPONENT, RIGHT
**
1651  10.0     -0.1000005E+02  -10.0
1652  10.0     -0.9009584E+01  -10.0
1653  10.0     -0.8019119E+01  -10.0
                  .
                  .
                  .
**
**   DEFINE NODE SETS FOR FILL OPERATION
**
*NSET, NSID =   10
   1,    2,    3,    4,    5,    6,    7,    8,    9,   10
  11,   12,   13,   14,   15
*NSET, NSID =   20
 151,  152,  153,  154,  155,  156,  157,  158,  159,  160
 161,  162,  163,  164,  165
*NSET, NSID =   30
 301,  302,  303,  304,  305,  306,  307,  308,  309,  310
 311,  312,  313,  314,  315
                    .
                    .
                    .
*nset, nsid = 620
 7098, 7113, 7128, 7143, 7158, 7166, 7174, 7182, 7190, 7198
 7206, 7214, 7229, 7244, 7259
*nset, nsid = 630
 8098, 8113, 8128, 8143, 8158, 8166, 8174, 8182, 8190, 8198
 8206, 8214, 8229, 8244, 8259
*nset, nsid = 640
 9098, 9113, 9128, 9143, 9158, 9166, 9174, 9182, 9190, 9198
 9206, 9214, 9229, 9244, 9259
**
```

```
** EQUIVALENCE NODES TO JOIN SEPARATE MODEL DOMAINS
**
*EQUIVALENCE
 500,  20
 510,  40
 520,  60
 530,  80
 540, 100
 600, 120
 610, 140
 620, 160
 630, 180
 640, 200
**
** GENERATE NODE PLANES
**
*NFILL
 10,  20,    9, 15
 30,  40,    9, 15
 50,  60,    9, 15
 70,  80,    9, 15
 90, 100,    9, 15
110, 120,    9, 15
130, 140,    9, 15
150, 160,    9, 15
170, 180,    9, 15
190, 200,    9, 15
**
**       NODES, ELEMENTS, AND NODE SETS FOR DELAMINATED VERTICAL
**       COMPONENT OBTAINED FROM THE FOLLOWING *MODEL GENERATION
**       STATEMENT:
**
**                   *MODEL GENERATION
**                    4,5,10.0,10.0,5,5,5,5
**                    1.5,1,1,1,1,1,1
**                    0.0, 0.0, 1.0
**                    0.0, 1.0, 0.0
**                    1.0, 0.0, 0.0
**                    1.0,10.0
**                    1,6,250,5000,5000
**                    1.0,9.875
**                    2,6,255,6000,6000
**                    1.0,9.750
**                    3,6,260,7000,7000
**                    1.0,9.625
**                    4,6,265,8000,8000
**                    1.0,9.50
**                    5,6,270,9000,9000
**
*NODE
   5001   0.1000000E+02   0.0000000E+00   0.0000000E+00
   5002   0.1000000E+02   0.1333340E+01   0.0000000E+00
   5003   0.1000000E+02   0.1333340E+01   0.1333340E+01
   5004   0.1000000E+02   0.0000000E+00   0.1333340E+01
```

```
   5005   0.1000000E+02  -0.1333340E+01   0.1333340E+01
                               .
                               .
                               .
   9268   0.9500000E+01   0.1000005E+02   0.4894511E+01
   9269   0.9500000E+01   0.1000005E+02   0.6038188E+01
   9270   0.9500000E+01   0.1000005E+02   0.7028653E+01
   9271   0.9500000E+01   0.1000005E+02   0.8019119E+01
   9272   0.9500000E+01   0.1000005E+02   0.9009584E+01
   9273   0.9500000E+01   0.1000005E+02   0.1000005E+02
**
** NODE SETS TO DEFINE DELAMINATION PLANES
**
*NSET, NSID =     255
   6001    6002    6003    6004    6005    6006    6007    6008
   6009    6010    6011    6012    6013    6014    6015    6016
   6017    6018    6019    6020    6021    6022    6023    6024
   6025    6026    6027    6028    6029    6030    6031    6032
   6033    6034    6035    6036    6037    6038    6039    6040
   6041
*NSET, NSID =     260
   7001    7002    7003    7004    7005    7006    7007    7008
   7009    7010    7011    7012    7013    7014    7015    7016
   7017    7018    7019    7020    7021    7022    7023    7024
   7025    7026    7027    7028    7029    7030    7031    7032
   7033    7034    7035    7036    7037    7038    7039    7040
   7041
*NSET, NSID =     265
   8001    8002    8003    8004    8005    8006    8007    8008
   8009    8010    8011    8012    8013    8014    8015    8016
   8017    8018    8019    8020    8021    8022    8023    8024
   8025    8026    8027    8028    8029    8030    8031    8032
   8033    8034    8035    8036    8037    8038    8039    8040
   8041
**
**   DEFINE GENERATOR ELEMENTS
**
** top horizontal component
**
*ELEMENT, LAYUP = 1
   1,  301,  316,  317,  302,    1,   16,   17,    2
*ELEMENT, LAYUP = 2
 300,  601,  616,  617,  602,  301,  316,  317,  302
*ELEMENT, LAYUP = 3
 600,  901,  916,  917,  902,  601,  616,  617,  602
*ELEMENT, LAYUP = 4
 900, 1201, 1216, 1217, 1202,  901,  916,  917,  902
**
** bottom horizontal component
**
*ELEMENT, LAYUP = 1
1200, 1801, 1816, 1817, 1802, 1501, 1516, 1517, 1502
*ELEMENT, LAYUP = 2
1500, 2101, 2116, 2117, 2102, 1801, 1816, 1817, 1802
```

```
*ELEMENT, LAYUP = 3
1800, 2401, 2416, 2417, 2402, 2101, 2116, 2117, 2102
*ELEMENT, LAYUP = 4
2100, 2701, 2716, 2717, 2702, 2401, 2416, 2417, 2402
**
**   ELEMENT GENERATION
**
*ELGEN, ELID = 10
   1, 10,15,1, 14,1,10,,,
 300, 10,15,1, 14,1,10,,,
 600, 10,15,1, 14,1,10,,,
 900, 10,15,1, 14,1,10,,,
1200, 10,15,1, 14,1,10,,,
1500, 10,15,1, 14,1,10,,,
1800, 10,15,1, 14,1,10,,,
2100, 10,15,1, 14,1,10,,,
**
** ELEMENTS IN VERTICAL COMPONENT FROM PRIOR *MODEL GENERATION RUN
**
*ELEMENT, LAYUP = 5, ORIENTATION = 1
   5001    6001    6004    6003    6002    5001    5004    5003    5002
   5002    6006    6005    6004    6001    5006    5005    5004    5001
   5003    6007    6006    6001    6008    5007    5006    5001    5008
   5004    6008    6001    6002    6009    5008    5001    5002    5009
   5005    6002    6003    6011    6010    5002    5003    5011    5010
                                    .
                                    .
                                    .
   8239    9252    9253    9268    9267    8252    8253    8268    8267
   8240    9253    9254    9269    9268    8253    8254    8269    8268
   8241    9254    9255    9270    9269    8254    8255    8270    8269
   8242    9255    9256    9271    9270    8255    8256    8271    8270
   8243    9256    9257    9272    9271    8256    8257    8272    8271
   8244    9257    9258    9273    9272    8257    8258    8273    8272
**
** RELAX TOLERANCE OF DEFORMED GEOMETRY CHECKS
**
*DEFORMED GEOMETRY
20.0
**
** LAMINATE DESCRIPTION
**
*LAYER, LAYUP = 1
 1, 0.0416,  90.0
 1, 0.0208,  45.0
 1, 0.0208, -45.0
*LAYER, LAYUP = 2
 1, 0.0208,  45.0
 1, 0.0208, -45.0
 1, 0.1456,   0.0
*LAYER, LAYUP = 3
 1, 0.1456,   0.0
 1, 0.0208, -45.0
 1, 0.0208,  45.0
```

```
*LAYER, LAYUP = 4
 1, 0.0208, -45.0
 1, 0.0208,  45.0
 1, 0.0416,  90.0
**
*LAYER, LAYUP = 5
 1, 0.0312,   0.0
 1, 0.0208,  45.0
 1, 0.0208, -45.0
 1, 0.0260,  90.0
 1, 0.0260,  45.0
 1, 0.0260, -45.0
*LAYER, LAYUP = 6
 1, 0.0312,   0.0
 1, 0.0260,  45.0
 1, 0.0260, -45.0
 1, 0.0208,   0.0
*LAYER, LAYUP = 7
 1, 0.0208,   0.0
 1, 0.0260, -45.0
 1, 0.0260,  45.0
 1, 0.0312,   0.0
*LAYER, LAYUP = 8
 1, 0.0260, -45.0
 1, 0.0260,  45.0
 1, 0.0260,  90.0
 1, 0.0208, -45.0
 1, 0.0208,  45.0
 1, 0.0312,   0.0
**
**   MATERIAL DEFINITION
**
*MATERIAL, MATID = 1
  2.48E6,2.48E6,0.71E6,0.3E6,0.3E6,0.99E6
  0.28,0.28,0.243
*FAILURE CRITERIA, FCID = 1
 MAX-STRESS
 21.2E3, 47.8E3, 21.2E3, 47.8E3, 21.2E3, 21.0E3
 1.65E3, 1.65E3, 7.0E3
*DAMAGE LAW, DLID = 1
 NULL
**
**  ESTABLISH LOCAL COORDINATE SYSTEM
**
*ORIENTATION
 1,  0.0, 1.0, 0.0,  0.0, 0.0, 1.0
**
*BOUNDARY2
  10, 1, 1
  30, 1, 1
  50, 1, 1
  70, 1, 1
  90, 1, 1
*BOUNDARY2
```

```
 110, 1, 3
 130, 1, 3
 150, 1, 3
 170, 1, 3
 190, 1, 3
**
*CLOAD
 136, 3,  -50.0
 137, 3, -100.0
 138, 3, -100.0
 139, 3, -100.0
 140, 3, -100.0
 141, 3, -100.0
 142, 3, -100.0
 143, 3, -100.0
 144, 3, -100.0
 145, 3, -100.0
 146, 3, -100.0
 147, 3, -100.0
 148, 3, -100.0
 149, 3, -100.0
 150, 3,  -50.0
 151, 3,  -50.0
 152, 3, -100.0
 153, 3, -100.0
 154, 3, -100.0
 155, 3, -100.0
 156, 3, -100.0
 157, 3, -100.0
 158, 3, -100.0
 159, 3, -100.0
 160, 3, -100.0
 161, 3, -100.0
 162, 3, -100.0
 163, 3, -100.0
 164, 3, -100.0
 165, 3,  -50.0
*ENDDTA
```

7.2.2 Solution Timing Summary for Linear Buckling Analysis

```
@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@
@@                                               @@
@@        U.S. ARMY RESEARCH LABORATORY          @@
@@                                               @@
@@                R E S T R A N                  @@
@@                                               @@
@@     RESIDUAL STRENGTH ANALYSIS OF IMPACT      @@
@@         DAMAGED COMPOSITE LAMINATES           @@
@@                                               @@
@@                 VERSION 1.0                   @@
@@                                               @@
@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@
```

```
          M O D E L   S I Z E   P A R A M E T E R S

             NUMBER OF ELEMENTS =        2096
             NUMBER OF NODES    =        2865
             DEGREES OF FREEDOM =        8595
             SYSTEM BANDWIDTH   =        1005

     TOTAL RAM MEMORY AVAILABLE =    14500000 WORDS

     TOTAL RAM MEMORY REQUIRED  =     9232360 WORDS

    ******************************************
    **                                      **
    ** TIMING SUMMARY OF PROGRAM EXECUTION **
    **                                      **
    ******************************************

  RESTRAN PREFACE OPERATIONS:  24.5313 SECONDS

    @@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@
    @@@                                     @@@
    @@@ BEGINNING LINEAR BUCKLING ANALYSIS @@@
    @@@                                     @@@
    @@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@

ELASTIC STIFFNESS ASSEMBLY TIME:       42.2373 SECONDS

BOUNDARY CONSTRAINT PROCESSING TIME:   11.7441 SECONDS

LINEAR SOLUTION TIME:                     2 MINUTES 33.4424 SECONDS

STRESS RECOVERY TIME:                  37.7783 SECONDS

DIFFERENTIAL STIFFNESS ASSEMBLY TIME:  14.0586 SECONDS

BOUNDARY CONSTRAINT PROCESSING TIME:   11.5098 SECONDS

BOUNDARY CONSTRAINT PROCESSING TIME:   11.9873 SECONDS

EIGENVALUE EXTRACTION TIME:            1 HOURS   53 MINUTES 45.6260 SECONDS

GRAPHICAL OUTPUT GENERATION TIME:      12.9336 SECONDS

     TOTAL JOB EXECUTION TIME:      1 HOURS   58 MINUTES 44.2803 SECONDS
```

7.2.3 Linear Buckling Solution Output File

```
@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@
@@                                          @@
@@      U.S. ARMY RESEARCH LABORATORY       @@
@@                                          @@
@@              R E S T R A N               @@
@@                                          @@
@@  RESIDUAL STRENGTH ANALYSIS OF IMPACT    @@
@@      DAMAGED COMPOSITE LAMINATES         @@
@@                                          @@
@@              VERSION 1.0                 @@
@@                                          @@
@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@

*** MESSAGE: ELEMENT NODE ORDER IS BEING CONVERTED TO RESTRAN FORMAT

      ######################################
      ###                                ###
      ### M O D E L   D E F I N I T I O N ###
      ###                                ###
      ######################################

              J O B   H E A D I N G:
              ---------------------

 MODEL OF A COMPOSITE CASING

           S O L U T I O N   C O N T R O L:
           -------------------------------

 S O L U T I O N   A L G O R I T H M:

   LINEAR BUCKLING ANALYSIS

 M E M O R Y   A L L O C A T I O N:

      BANDWIDTH STORAGE MODE

    * BANDWIDTH MINIMIZATION WILL BE PERFORMED

           P R O G R A M   O P T I O N S:
           -----------------------------

    * EXTENDED JOB EXECUTION STATISTICS WILL BE PRINTED

      R E Q U E S T E D   P R I N T   O P T I O N S:
      ---------------------------------------------

 * PRINT DISPLACEMENTS AFTER CONVERGENCE AT THE iTH LOAD INCREMENT
```

```
* PRINT BUCKLING MODE SHAPE CORRESPONDING TO THE LOWEST BUCKLING
  LOAD MULTIPLIER AT THE iTH ITERATION


                     G R A P H I C S:
                     ----------------


   @ GRAPHICAL OUTPUT IN MATHEMATICA FORMAT WILL BE GENERATED

   @ BOTH UNDEFORMED AND DEFORMED GEOMETRY IS REQUESTED
     DEFORMATION/MODE MAGNIFICATION FACTOR = 0.1000E+01

   @ RESTRAN GRAPHICS CONTAINED IN FILE: restran.grf


                N O D E   D E F I N I T I O N S:
                --------------------------------

      NODE ID         X           Y           Z

           1      0.000E+00   -.100E+02   0.100E+02
           2      0.000E+00   -.901E+01   0.100E+02
           3      0.000E+00   -.802E+01   0.100E+02
           4      0.000E+00   -.703E+01   0.100E+02
           5      0.000E+00   -.604E+01   0.100E+02

                                  .
                                  .
                                  .
        9268      0.950E+01   0.100E+02   0.489E+01
        9269      0.950E+01   0.100E+02   0.604E+01
        9270      0.950E+01   0.100E+02   0.703E+01
        9271      0.950E+01   0.100E+02   0.802E+01
        9272      0.950E+01   0.100E+02   0.901E+01


               S P E C I F I E D   N O D E   S E T S:
               --------------------------------------


                     SET ID    10:

      1       2       3       4       5       6       7       8       9      10
     11      12      13      14      15
                              .
                              .
                              .

                     SET ID   265:

   8001    8002    8003    8004    8005    8006    8007    8008    8009    8010
   8011    8012    8013    8014    8015    8016    8017    8018    8019    8020
```

8021	8022	8023	8024	8025	8026	8027	8028	8029	8030
8031	8032	8033	8034	8035	8036	8037	8038	8039	8040
8041									

E L E M E N T   D E F I N I T I O N S:

ELEMENT	N1	N2	N3	N4	N5	N6	N7	N8
1	301	316	317	302	1	16	17	2
2	316	331	332	317	16	31	32	17
3	331	346	347	332	31	46	47	32
4	346	361	362	347	46	61	62	47
5	361	376	377	362	61	76	77	62
				.				
				.				
				.				
8240	9253	9268	9269	9254	8253	8268	8269	8254
8241	9254	9269	9270	9255	8254	8269	8270	8255
8242	9255	9270	9271	9256	8255	8270	8271	8256
8243	9256	9271	9272	9257	8256	8271	8272	8257
8244	9257	9272	1365	1364	8257	8272	1065	1064

E L E M E N T   S E T S:

ELEMENT SET ID 10:

2	3	4	5	6	7	8	9	10	11
12	13	14	15	16	17	18	19	20	21
22	23	24	25	26	27	28	29	30	31
				.					
				.					
				.					
2218	2219	2220	2221	2222	2223	2224	2225	2226	2227
2228	2229	2230	2231	2232	2233	2234	2235	2236	2237
2238	2239								

M A T E R I A L   P R O P E R T Y   D A T A:

MATERIAL ID 1

E1 = 0.248E+07   G23 = 0.300E+06   MU23 = 0.28000
E2 = 0.248E+07   G13 = 0.300E+06   MU13 = 0.28000
E3 = 0.710E+06   G12 = 0.990E+06   MU12 = 0.24300

ASSOCIATED FAILURE LAW:

MAXIMUM STRESS CRITERION

XT = 0.212E+05   XC = 0.478E+05   YT = 0.212E+05

YC = 0.478E+05	ZT = 0.212E+05	ZC = 0.210E+05
R = 0.165E+04	S = 0.165E+04	T = 0.700E+04

ASSOCIATED DAMAGE LAW:

NULL POST-FAILURE PROPERTIES

MATERIAL ID 2

E1 = 0.100E+06	G23 = 0.385E+05	MU23 = 0.30000
E2 = 0.100E+06	G13 = 0.385E+05	MU13 = 0.30000
E3 = 0.100E+06	G12 = 0.385E+05	MU12 = 0.30000

ASSOCIATED FAILURE LAW:

MAXIMUM STRESS CRITERION

XT = 0.212E+05	XC = 0.478E+05	YT = 0.212E+05
YC = 0.478E+05	ZT = 0.212E+05	ZC = 0.210E+05
R = 0.165E+04	S = 0.165E+04	T = 0.700E+04

ASSOCIATED DAMAGE LAW:

NULL POST-FAILURE PROPERTIES

M A T E R I A L   L A Y E R   D E F I N I T I O N S

LAYUP ID = 1
NUMBER OF PLIES = 3

MATERIAL ID	PLY THICKNESS	ORIENTATION
1	0.04160	90.00000
1	0.02080	45.00000
1	0.02080	-45.00000

.
.
.

LAYUP ID = 88
NUMBER OF PLIES = 6

MATERIAL ID	PLY THICKNESS	ORIENTATION
2	0.02600	-45.00000
2	0.02600	45.00000
2	0.02600	90.00000
2	0.02080	-45.00000
2	0.02080	45.00000
2	0.03120	0.00000

M A T E R I A L   C O O R D I N A T E   S Y S T E M S

ID	V11	V12	V13	V21	V22	V23
1	0.000E+00	0.100E+01	0.000E+00	0.000E+00	0.000E+00	0.100E+01

B O U N D A R Y   C O N D I T I O N S:

SINGLE POINT CONSTRAINTS:

NODE	DOF1	TO DOF2
1	1	3
2	1	3
3	1	3
4	1	3
5	1	3
	.	
	.	
	.	
2711	1	3
2712	1	3
2713	1	3
2714	1	3
2715	1	3

APPLIED CONCENTRATED FORCES:

NODE	DIRECTION	MAGNITUDE
1651	2	0.500E+01
5099	2	0.100E+02
5100	2	0.100E+02
5101	2	0.100E+02
	.	
	.	
	.	
9269	2	-.100E+02
9270	2	-.100E+02
9271	2	-.100E+02
9272	2	-.100E+02
1365	2	-.500E+01

FORCE/MOMENT RESULTANTS AT ORIGIN:

|R| = 0.000E+00 |M| = 0.000E+00

Rx = 0.000E+00 Ry = 0.000E+00 Rz = 0.000E+00

```
Mx = 0.000E+00  My = 0.000E+00  Mz = 0.000E+00

Xo = 0.000E+00  Yo = 0.000E+00  Zo = 0.000E+00


*** E N D  O F  M O D E L  D E F I N I T I O N ***


     M O D E L  S I Z E  P A R A M E T E R S

        NUMBER OF ELEMENTS =        2096
        NUMBER OF NODES    =        2865
        DEGREES OF FREEDOM =        8595
        SYSTEM BANDWIDTH   =        1005

 ####################################################
 ##                                                ##
 ## L I N E A R  B U C K L I N G  A N A L Y S I S ##
 ##                                                ##
 ####################################################

    CONVERGED EIGENVALUES:

     NUMBER      LAMBDA

        1    0.1782068E+03
        2    0.1763134E+03
        3    0.8268226E+02
        4    0.7179661E+02

               BUCKLING MODE SHAPE
               (SCALE = 0.718E+02)

NODE        U            V            W
   1  0.00000E+00  0.00000E+00  0.00000E+00
   2  0.00000E+00  0.00000E+00  0.00000E+00
   3  0.00000E+00  0.00000E+00  0.00000E+00
   4  0.00000E+00  0.00000E+00  0.00000E+00
   5  0.00000E+00  0.00000E+00  0.00000E+00
                       .
                       .
                       .
9268  0.67520E+00  0.41156E-01  -.27246E-01
9269  0.52133E+00  0.32320E-01  -.31375E-01
9270  0.37198E+00  0.22749E-01  -.33309E-01
9271  0.21898E+00  0.12126E-01  -.31172E-01
9272  0.83349E-01  0.21711E-02  -.25241E-01


@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@
@@                                                        @@
@@  B U C K L I N G  A N A L Y S I S  C O M P L E T E D  @@
@@                                                        @@
@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@
```

A graphical depiction of the deformed shape is shown in Figure 31.

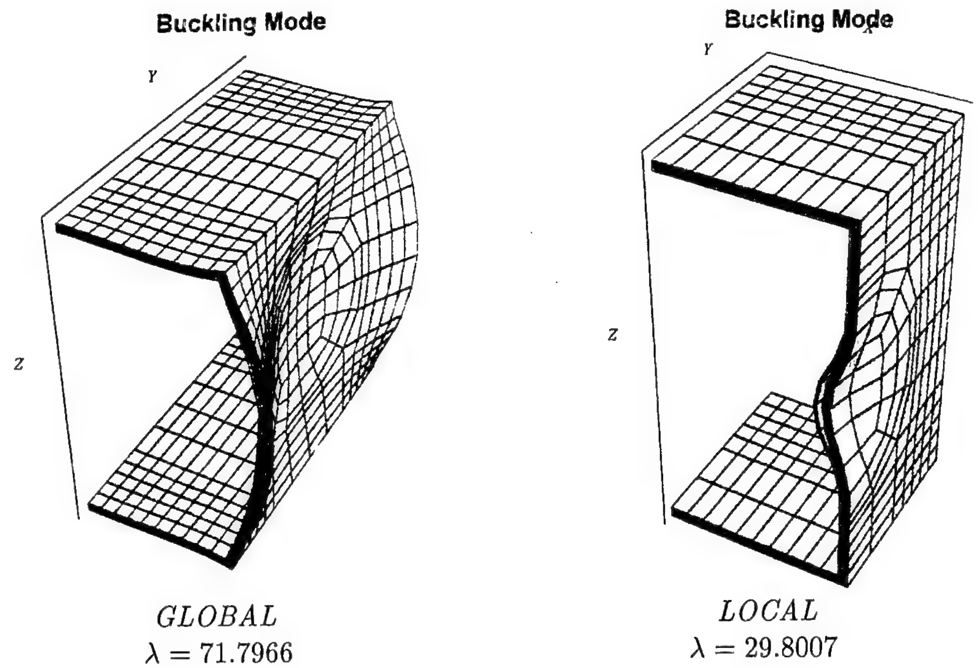


Figure 31. Global and local buckling mode shapes.

The presence of the delaminations at the specified ply interfaces is thus shown to reduce the critical buckling load by 66%.

7.3 Residual Strength Analysis

To illustrate the execution of a residual strength analysis, a model of an elastically supported laminated composite face plate is analyzed. Due to the lack of data of sufficient resolution regarding the spatial distribution of specific internal damage modes and subsequent experimental determination of residual strength, a specific state of internal damage is assumed for the following example.

The geometry and applied loading is shown in Figure 32. Material properties were selected as S2-Glass/3501 Epoxy tape with a nominal ply thickness of 0.0052 in. The material elastic properties are given by

$$\begin{array}{lllllllll} E_1 & = & 7.150E6 & E_2 & = & 2.13E6 & E_3 & = & 2.13E6 \\ G_{23} & = & 0.71E6 & G_{13} & = & 0.98E6 & G_{12} & = & 0.98E6 \\ \nu_{23} & = & 0.499 & \nu_{13} & = & 0.306 & \nu_{12} & = & 0.296 \end{array}$$

with strengths given by

$$\begin{array}{lllllllll} X_{Ten} & = & 2.43E5 & X_{Comp} & = & 1.77E5 & Y_{Ten} & = & 7.0E3 \\ Y_{Comp} & = & 3.06E4 & Z_{Ten} & = & 7.03E3 & Z_{Comp} & = & 3.5E4 \\ R & = & 1.7E4 & S & = & 1.57E4 & T & = & 1.57E4 \end{array}$$

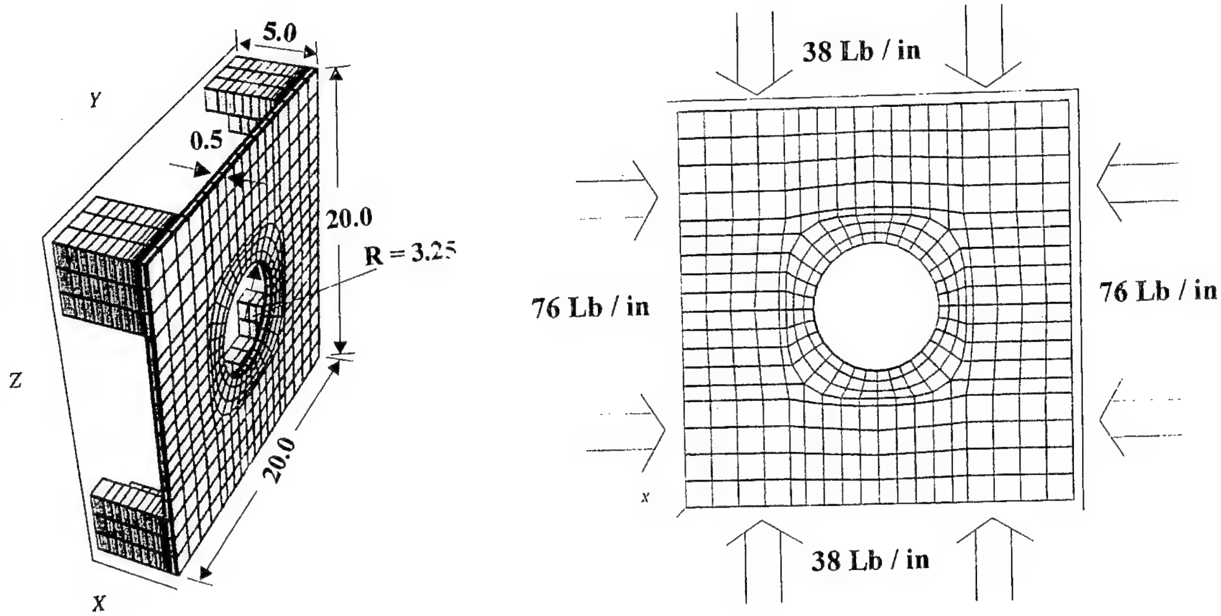


Figure 32. Geometry and loading of an elastically supported composite plate.

The face plate is composed of 96 plies and is assumed to have been subjected to a normal impact leading to an inner circular region where all properties are effectively zero and the region is modeled as an open hole. The laminate was divided into three layers, where '|' indicates a layer interface. The layup is given by $[\pm 45/0_6 \mid \pm 45_5/90_6 \pm 45_5/0_6 \pm 45_2/0_8/\mp 45_2/0_6 \mp 45_5/90_6/\mp 45_5 \mid 0_6/\mp 45]$. A residual strength analysis is performed which is selected to evaluate material failure with nonlinear material properties. Secant moduli were computed for the inplane Young's moduli as

$$E_i' = E_i(1.0 - 0.75\left|\frac{\sigma_{ii}}{\alpha_i}\right|), \tag{20}$$

and for shear moduli as

$$G_{ij}' = G_{ij}(1.0 - 0.5\left|\frac{\tau_{ii}}{\beta_i}\right|), \tag{21}$$

where α and β are taken as material strengths given by

$$\begin{array}{llllll} \alpha_1 = & 2.43E5 & \alpha_2 = & 7.0E3 & \alpha_3 = & 7.03E3 \\ \beta_1 = & 1.7E4 & \beta_2 = & 1.57E4 & \beta_3 = & 1.57E4 \end{array}$$

The maximum reduction in Young's moduli was limited to 25% of the initial values and 50% for the shear moduli. For this example, the maximum stress failure criteria is used with a nonspecific damage law that assigns zero modulus to plies experiencing failure without regard to failure mode. Abbreviated input and output files are shown, followed by a graphical presentation of the progression of failure.

7.3.1 Input Data File for a Residual Strength Problem

```
*HEADING
 MODEL OF AN ELASTICALLY SUPPORTED COMPOSITE PLATE WITH AN OPEN HOLE
**
** [45/-45/0_6|(45/-45)_5/90_6/(45/-45)_5/0_6/(45/-45)_2/0_8/
** (-45/45)_2/0_6/(-45/45)_5/90_6/(-45/45)_5|0_6/-45/45]
**
**           LAYUP TAPE THICKNESS = 0.0052''
**
*SOLUTION, METH = MTL
 6,15,1.05
**ECHO
**PREPASS
**NODE PRINT
**  D,M
**PLY FAILURE PRINT
*NONLINEAR MATERIAL
 10, 0.01, 2.0
3
 2.43E5, 1.77E5
 7.0E3, 3.06E4
 7.0E3, 3.5E4
 1.7E4, 1.57E4, 1.57E4
*parameter lapfail
*parameter directory = current
*MEMORY ALLOCATION
 BAND
*GRAPHICS, format = mathematica
 2, 2, 1.0
**
**   NODE DEFINITIONS
**
*NODE
**
** elastic supports
**
   1   5.0        -10.0         10.0
   2   5.8        -10.0         10.0
   3   6.3        -10.0         10.0
   4   6.8        -10.0         10.0
   5   7.3        -10.0         10.0
                    .
                    .
                    .
4708   8.7         -8.6396     -5.9189
4709   8.9875      -8.6396     -5.9189
4710   9.1875      -8.6396     -5.9189
4711   9.3375      -8.6396     -5.9189
4712   9.4375      -8.6396     -5.9189
4713   9.5         -8.6396     -5.9189
**
**      NODES, ELEMENTS, AND NODE SETS FOR DELAMINATED VERTICAL
**      COMPONENT OBTAINED FROM THE FOLLOWING *MODEL GENERATION
```

```
**      STATEMENT:
**
**                  *MODEL GENERATION
**                   8,4,10.0,10.0,4,4,4,4
**                   2.0,1,1,1,0,1
**                   0.0, 0.0, 1.0
**                   0.0, 1.0, 0.0
**                   1.0, 0.0, 0.0
**                   1.0,10.0
**                   1,5,250,5000,5000
**                   0.95,9.875
**                   2,6,255,7000,7000
**                   0.85,9.625
**                   3,7,265,11000,11000
**                   0.8,9.50
**                   4,8,270,13000,13000
**
*NODE
   5114   0.1000000E+02   0.3250000E+01   0.0000000E+00
   5115   0.1000000E+02   0.3198816E+01   0.5745243E+00
   5116   0.1000000E+02   0.3040097E+01   0.1149049E+01
   5117   0.1000000E+02   0.2755322E+01   0.1723573E+01
   5118   0.1000000E+02   0.2298097E+01   0.2298097E+01
                              .
                              .
                              .
  13517   0.9500000E+01   0.1000000E+02   0.4558612E+01
  13518   0.9500000E+01   0.1000000E+02   0.5918959E+01
  13519   0.9500000E+01   0.1000000E+02   0.7279306E+01
  13520   0.9500000E+01   0.1000000E+02   0.8639653E+01
  13521   0.9500000E+01   0.1000000E+02   0.1000000E+02
**
**   DEFINE GENERATOR ELEMENTS
**
*ELEMENT, LAYUP = 4, ELSET = 300
**
** top front elastic support
**
   1,   401,   402,  902,  901,    1,    2,  102,  101
 101,   501,   502,  802,  801,  401,  402,  902,  901
 201,   601,   602,  702,  701,  501,  502,  802,  801
 301,   901,   902, 1002, 1001,  101,  102,  202,  201
 401,  1001,  1002, 1102, 1101,  201,  202,  302,  301
**
** top back elastic support
**
 501,  1601,  1602, 1702, 1701, 1201, 1202, 1302, 1301
 601,  1701,  1702, 1802, 1801, 1301, 1302, 1402, 1401
 701,  1801,  1802, 1902, 1901, 1401, 1402, 1502, 1501
 801,  2001,  2002, 2102, 2101, 1801, 1802, 1902, 1901
 901,  2201,  2202, 2302, 2301, 2001, 2002, 2102, 2101
**
** bottom front elastic support
**
```

```
1001,  3601,  3602, 3702, 3701, 4001, 4002, 4102, 4101
1101,  4001,  4002, 4102, 4101, 4401, 4402, 4502, 4501
1201,  4401,  4402, 4502, 4501, 4601, 4602, 4702, 4701
1301,  3701,  3702, 3802, 3801, 4101, 4102, 4202, 4201
1401,  3801,  3802, 3902, 3901, 4201, 4202, 4302, 4301
**
** bottom back elastic support
**
1501,  2401,  2402, 2502, 2501, 2801, 2802, 2902, 2901
1601,  2501,  2502, 2602, 2601, 2901, 2902, 3002, 3001
1701,  2601,  2602, 2702, 2701, 3001, 3002, 3102, 3101
1801,  3001,  3002, 3102, 3101, 3201, 3202, 3302, 3301
1901,  3201,  3202, 3302, 3301, 3401, 3402, 3502, 3501
**
**   ELEMENT GENERATION FOR ELASTIC SUPPORTS
**
*ELGEN, ELID = 300
   1, 12,1,1,,,,,,
 101, 12,1,1,,,,,,
       .
       .
       .
1801, 12,1,1,,,,,,
1901, 12,1,1,,,,,,
**
** CREATE NODE SETS FOR EQUIVALENCING
**
*NSET, NSID =  10
   13,   113,   213,   313,   413,   913,  1013,  1113
  513,   813,   613,   713
*NSET, NSID = 20
13300, 13319, 13338, 13357, 13299, 13318, 13337, 13356
13298, 13317, 13297, 13316
  .
  .
  .
*NSET, NSID = 70
 3413,  3513,  3213,  3313,  2813,  2913,  3013,  3113
 2413,  2513,  2613,  2713
*NSET, NSID = 80
13487, 13506, 13486, 13505, 13447, 13466, 13485, 13504
13446, 13465, 13484, 13503
**
** EQUIVALENCE NODE SETS
**
*EQUIVALENCE
 10, 20
 30, 40
 50, 60
 70, 80
**
** ELEMENTS IN VERTICAL COMPONENT FROM PRIOR *MODEL GENERATION RUN
**
*ELEMENT, LAYUP =          1 ORIENTATION =          1
```

```
   5129    7114    7115    7147    7146    5114    5115    5147    5146
   5130    7115    7116    7148    7147    5115    5116    5148    5147
   5131    7116    7117    7149    7148    5116    5117    5149    5148
                                   .
                                   .
                                   .
   5482    7499    7500    7519    7518    5499    5500    5519    5518
   5483    7500    7501    7520    7519    5500    5501    5520    5519
   5484    7501    7502    7521    7520    5501    5502    5521    5520
*ELEMENT, LAYUP =           2 ORIENTATION =           1
   7129   11114   11115   11147   11146    7114    7115    7147    7146
   7130   11115   11116   11148   11147    7115    7116    7148    7147
   7131   11116   11117   11149   11148    7116    7117    7149    7148
                                   .
                                   .
                                   .
   7482   11499   11500   11519   11518    7499    7500    7519    7518
   7483   11500   11501   11520   11519    7500    7501    7520    7519
   7484   11501   11502   11521   11520    7501    7502    7521    7520
*ELEMENT, LAYUP =           3 ORIENTATION =           1
  11129   13114   13115   13147   13146   11114   11115   11147   11146
  11130   13115   13116   13148   13147   11115   11116   11148   11147
  11131   13116   13117   13149   13148   11116   11117   11149   11148
                                   .
                                   .
                                   .
  11482   13499   13500   13519   13518   11499   11500   11519   11518
  11483   13500   13501   13520   13519   11500   11501   11520   11519
  11484   13501   13502   13521   13520   11501   11502   11521   11520
**
** EXCLUDE ELEMENTS IN ELASTIC SUPPORTS
**
*EXCLUDE ELEMENT
 300
**
** RELAX TOLERANCE OF DEFORMED GEOMETRY CHECKS
**
*DEFORMED GEOMETRY
20.0
**
** LAMINATE DESCRIPTION
**
*LAYER, LAYUP = 1
 1, 0.0052,  45.0
 1, 0.0052, -45.0
 1, 0.0312,   0.0
*LAYER, LAYUP = 2
 1, 0.0260,  45.0
 1, 0.0260, -45.0
 1, 0.0312,  90.0
 1, 0.0260,  45.0
 1, 0.0260, -45.0
 1, 0.0312,   0.0
 1, 0.0104, -45.0
```

```
 1, 0.0104,  45.0
 1, 0.0416,   0.0
 1, 0.0104, -45.0
 1, 0.0104,  45.0
 1, 0.0312,   0.0
 1, 0.0260, -45.0
 1, 0.0260,  45.0
 1, 0.0312,  90.0
 1, 0.0260, - 45.0
 1, 0.0260,  45.0
*LAYER, LAYUP=3
 1, 0.0312,   0.0
 1, 0.0052, - 45.0
 1, 0.0052,  45.0
*LAYER, LAYUP = 4
 2, 1.0, 0.0
**
**   MATERIAL DEFINITIONS
**
** COMPOSITE PLY PROPERTIES (S2-GLASS/3501 EPOXY)
**
*MATERIAL, MATID = 1
 7.150E6, 2.13E6, 2.13E6, 0.98E6, 0.71E6, 0.98E6
  0.306, 0.499, 0.296
*FAILURE CRITERIA, FCID = 1
 MAX-STRESS
 2.43E5, 1.77E5, 7.0E3, 3.06E4, 7.0E3, 3.5E4
 1.7E4, 1.57E4, 1.57E4
*DAMAGE LAW, DLID = 1
 NULL
**
** MEOTALLIC PROPERTIES
**
*MATERIAL, MATID = 2
 1.0E8, 1.0E8, 1.0E8, 0.3846E8, 0.3846E8, 0.3846E8
  0.3, 0.3, 0.3
*FAILURE CRITERIA, FCID = 2
 MAX-STRESS
 5.3E4, 2.2E5, 5.3E4, 2.2E5, 5.3E4, 2.2E6
 4.8E4, 4.8E4, 4.8E4
*DAMAGE LAW, DLID = 2
 NULL
**
**  ESTABLISH LOCAL COORDINATE SYSTEM
**
*ORIENTATION
 1,  0.0, 1.0, 0.0,  0.0, 0.0, 1.0
**
** NODE SET DEFINITION FOR BOUNDARY CONSTRAINT INPUT
**
*NSET, NSID = 100
    1,  101,  201,  301,  401,  501,  601,  701,  801
  901, 1001, 1101
*NSET, NSID = 110
```

```
 1201, 1301, 1401, 1501, 1601, 1701, 1801, 1901, 2001
 2101, 2201, 2301
*NSET, NSID = 120
 2401, 2501, 2601, 2701, 2801, 2901, 3001, 3101, 3201
 3301, 3401, 3501
*NSET, NSID = 130
 3601, 3701, 3801, 3901, 4001, 4101, 4201, 4301, 4401
 4501, 4601, 4701
*BOUNDARY2
  100, 1, 3
  110, 1, 3
  120, 1, 3
  130, 1, 3
**
** biaxial y-z plane loading
**
*CLOAD
    5300, 3, -.1000000E+02
    5319, 3, -.1000000E+02
    5338, 3, -.1000000E+02
          .
          .
          .
   13519, 2,  -.2000000E+02
   13520, 2,  -.2000000E+02
   13521, 2,  -.2000000E+02
*ENDDATA
```

7.3.2 Output Data File for a Residual Strength Problem

```
@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@
@@                                          @@
@@      U.S. ARMY RESEARCH LABORATORY       @@
@@                                          @@
@@              R E S T R A N               @@
@@                                          @@
@@   RESIDUAL STRENGTH ANALYSIS OF IMPACT   @@
@@       DAMAGED COMPOSITE LAMINATES        @@
@@                                          @@
@@               VERSION 1.0                @@
@@                                          @@
@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@

*** MESSAGE: ELEMENT NODE ORDER IS BEING CONVERTED TO RESTRAN FORMAT

*** WARNING: MATERIAL ID      1 HAS A MIXED MODE FAILURE CRITERIA
             ASSOCIATED WITH A SINGLE MODE DAMAGE LAW (NULL).
             NULL ACCEPTED

*** WARNING: MATERIAL ID      2 HAS A MIXED MODE FAILURE CRITERIA
             ASSOCIATED WITH A SINGLE MODE DAMAGE LAW (NULL).
             NULL ACCEPTED
```

```
#######################################
###                                 ###
### M O D E L   D E F I N I T I O N ###
###                                 ###
#######################################
```

J O B   H E A D I N G:

MODEL OF AN ELASTICALLY SUPPORTED COMPOSITE PLATE

S O L U T I O N   C O N T R O L:

S O L U T I O N   A L G O R I T H M:

RESIDUAL STRENGTH ANALYSIS: MATERIAL FAILURE ONLY

I T E R A T I O N   P A R A M E T E R S:

MAXIMUM NUMBER OF GLOBAL ANALYSIS CYCLES = 6
NUMBER OF ITERATIONS TO CONVERGE MATERIAL FAILURE = 15

M E M O R Y   A L L O C A T I O N:

BANDWIDTH STORAGE MODE

* BANDWIDTH MINIMIZATION WILL BE PERFORMED

M A T E R I A L   B E H A V I O R:

NONLINEAR ELASTIC

LOCAL NONLINEAR SOLUTION PARAMETERS:

MAXIMUM NUMBER OF ITERATIONS = 10
MATERIAL CONVERGENCE TOLERANCE = 0.100E-01
BUCKLING CONVERGENCE TOLERANCE = 0.200E+01

P R O G R A M   O P T I O N S:

* ELEMENTS ASSOCIATED WITH LOAD APPLICATION POINTS
  ARE EXCLUDED FROM EXHIBITING FAILURE

* EXTENDED JOB EXECUTION STATISTICS WILL BE PRINTED

* THE FOLLOWING ELEMENT SETS ARE EXCLUDED FROM FAILURE PREDICTION

300

REQUESTED PRINT OPTIONS:

* PRINT ELEMENT PLY FAILURE STATISTICS

GRAPHICS:

@ GRAPHICAL OUTPUT IN MATHEMATICA FORMAT WILL BE GENERATED

@ BOTH UNDEFORMED AND DEFORMED GEOMETRY IS REQUESTED
DEFORMATION/MODE MAGNIFICATION FACTOR = 0.1000E+01

@ RESTRAN GRAPHICS CONTAINED IN FILE: restran.grf

NODE DEFINITIONS:

NODE ID	X	Y	Z
1	0.500E+01	-.100E+02	0.100E+02
2	0.580E+01	-.100E+02	0.100E+02
3	0.630E+01	-.100E+02	0.100E+02
4	0.680E+01	-.100E+02	0.100E+02
5	0.730E+01	-.100E+02	0.100E+02
		.	
		.	
		.	
13517	0.950E+01	0.100E+02	0.456E+01
13518	0.950E+01	0.100E+02	0.592E+01
13519	0.950E+01	0.100E+02	0.728E+01
13520	0.950E+01	0.100E+02	0.864E+01
13521	0.950E+01	0.100E+02	0.100E+02

SPECIFIED NODE SETS:

SET ID 10:

13	113	213	313	413	913	1013	1113	513	813
613	713								

SET ID 20:

13300	13319	13338	13357	13299	13318	13337	13356	13298	13317
13297	13316								

.
.
.

ELEMENT DEFINITIONS:

ELEMENT	N1	N2	N3	N4	N5	N6	N7	N8

1	401	402	902	901	1	2	102	101
2	402	403	903	902	2	3	103	102
3	403	404	904	903	3	4	104	103
4	404	405	905	904	4	5	105	104
5	405	406	906	905	5	6	106	105
				.				
				.				
				.				
11480	13497	13516	13517	13498	11497	11516	11517	11498
11481	13498	13517	13518	13499	11498	11517	11518	11499
11482	13499	13518	13519	13500	11499	11518	11519	11500
11483	13500	13519	13520	13501	11500	11519	11520	11501
11484	13501	13520	13521	13502	11501	11520	11521	11502

E L E M E N T S E T S:

ELEMENT SET ID 300:

1 101 201 301 ... 1909 1910 1911 1912

M A T E R I A L P R O P E R T Y D A T A:

MATERIAL ID 1

E1 = 0.715E+07 G23 = 0.980E+06 MU23 = 0.30600
E2 = 0.213E+07 G13 = 0.710E+06 MU13 = 0.49900
E3 = 0.213E+07 G12 = 0.980E+06 MU12 = 0.29600

ASSOCIATED FAILURE LAW:

HOFFMAN CRITERION (3-D MIXED MODE)

ASSOCIATED DAMAGE LAW:

NULL POST-FAILURE PROPERTIES

MATERIAL ID 2

E1 = 0.100E+09 G23 = 0.385E+08 MU23 = 0.30000
E2 = 0.100E+09 G13 = 0.385E+08 MU13 = 0.30000
E3 = 0.100E+09 G12 = 0.385E+08 MU12 = 0.30000

ASSOCIATED FAILURE LAW:

MAXIMUM STRESS CRITERION

XT = 0.530E+05 XC = 0.220E+06 YT = 0.530E+05
YC = 0.220E+06 ZT = 0.530E+05 ZC = 0.220E+07
R = 0.480E+05 S = 0.480E+05 T = 0.480E+05

ASSOCIATED DAMAGE LAW:

NULL POST-FAILURE PROPERTIES

M A T E R I A L   L A Y E R   D E F I N I T I O N S

LAYUP ID = 1
NUMBER OF PLIES = 3

MATERIAL ID	PLY THICKNESS	ORIENTATION
1	0.00520	45.00000
1	0.00520	-45.00000
1	0.03120	0.00000

LAYUP ID = 2
NUMBER OF PLIES = 17

MATERIAL ID	PLY THICKNESS	ORIENTATION
1	0.02600	45.00000
1	0.02600	-45.00000
1	0.03120	90.00000
1	0.02600	45.00000
1	0.02600	-45.00000
1	0.03120	0.00000
1	0.01040	-45.00000
1	0.01040	45.00000
1	0.04160	0.00000
1	0.01040	-45.00000
1	0.01040	45.00000
1	0.03120	0.00000
1	0.02600	-45.00000
1	0.02600	45.00000
1	0.03120	90.00000
1	0.02600	0.00000
1	0.02600	45.00000

LAYUP ID = 3
NUMBER OF PLIES = 3

MATERIAL ID	PLY THICKNESS	ORIENTATION
1	0.03120	0.00000
1	0.00520	0.00000
1	0.00520	45.00000

LAYUP ID = 4
NUMBER OF PLIES = 1

MATERIAL ID	PLY THICKNESS	ORIENTATION
2	1.00000	0.00000

M A T E R I A L   C O O R D I N A T E   S Y S T E M S

ID	V11	V12	V13	V21	V22	V23
1	0.000E+00	0.100E+01	0.000E+00	0.000E+00	0.000E+00	0.100E+01

B O U N D A R Y   C O N D I T I O N S:

SINGLE POINT CONSTRAINTS:

NODE	DOF1	TO DOF2
1	1	1
101	1	1
201	1	1
301	1	1
	.	
	.	
	.	
4401	1	3
4501	1	3
4601	1	3
4701	1	3

APPLIED CONCENTRATED FORCES:

NODE	DIRECTION	MAGNITUDE
5300	3	-.100E+02
5319	3	-.100E+02
5338	3	-.100E+02
	.	
	.	
	.	
13519	2	-.200E+02
13520	2	-.200E+02
13521	2	-.200E+02

FORCE/MOMENT RESULTANTS AT ORIGIN:

|R| = 0.000E+00 |M| = 0.000E+00

Rx = 0.000E+00   Ry = 0.000E+00   Rz = 0.000E+00

Mx = 0.000E+00   My = 0.000E+00   Mz = 0.000E+00

Xo = 0.000E+00   Yo = 0.000E+00   Zo = 0.000E+00

*** E N D   O F   M O D E L   D E F I N I T I O N ***

M O D E L   S I Z E   P A R A M E T E R S

```
          NUMBER OF ELEMENTS =        1308
          NUMBER OF NODES    =        2208
          DEGREES OF FREEDOM =        6624
          SYSTEM BANDWIDTH   =        2574

   ################################################
   ##                                            ##
   ## B E G I N   F A I L U R E   A N A L Y S I S ##
   ##                                            ##
   ################################################

  @@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@
  @@                                                    @@
  @@     PRIMARY ANALYSIS CYCLE. PASS NUMBER     1      @@
  @@                                                    @@
  @@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@

         ####################################
         ###                              ###
         ### NONLINEAR SOLUTION PROCEDURE ###
         ###                              ###
         ####################################

<< GLOBAL ANALYSIS CYCLE:    1    ITERATION:     1 >>

* MATERIAL FAILURE LOAD CONVERGENCE FACTOR: 0.177E+03

<< GLOBAL ANALYSIS CYCLE:    1    ITERATION:     2 >>

* MATERIAL FAILURE LOAD CONVERGENCE FACTOR: 0.196E+00

<< GLOBAL ANALYSIS CYCLE:    1    ITERATION:     3 >>

* MATERIAL FAILURE LOAD CONVERGENCE FACTOR: 0.267E-01

<< GLOBAL ANALYSIS CYCLE:    1    ITERATION:     4 >>

* MATERIAL FAILURE LOAD CONVERGENCE FACTOR: 0.212E-01

<< GLOBAL ANALYSIS CYCLE:    1    ITERATION:     5 >>

* MATERIAL FAILURE LOAD CONVERGENCE FACTOR: 0.657E-02

      ************************************************
      *** NONLINEAR MATERIAL LOADS HAVE CONVERGED ***
      *** IN    5 ITERATIONS.                     ***
      ************************************************

     **************************************************
     **                                              **
     ** ALGORITHMIC PATH FOR MINIMUM LOAD INCREMENT **
     ** TO NEXT FAILURE: MATERIAL DEGRADATION        **
     **                                              **
```

```
**          MATERIAL          BUCKLING          **
**         ----------        ----------         **
** SCALE:   0.151E+03         EXCLUDED          **
**                                              **
**************************************************

          ********************************
          *                              *
          *   MATERIAL FAILURE ANALYSIS  *
          *                              *
          *  ITERATION NO.  SCALE FACTOR *
          *  -------------  ------------- *
          *        1          0.151E+03  *
          *                              *
          ********************************

* NUMBER OF ELEMENTS DEGRADED =        3

          ********************************
          *                              *
          *   MATERIAL FAILURE ANALYSIS  *
          *                              *
          *  ITERATION NO.  SCALE FACTOR *
          *  -------------  ------------- *
          *        2          0.151E+03  *
          *                              *
          ********************************

* NUMBER OF ELEMENTS DEGRADED =       14

          ********************************
          *                              *
          *   MATERIAL FAILURE ANALYSIS  *
          *                              *
          *  ITERATION NO.  SCALE FACTOR *
          *  -------------  ------------- *
          *        3          0.151E+03  *
          *                              *
          ********************************

* NUMBER OF ELEMENTS DEGRADED =        7

          ********************************
          *                              *
          *   MATERIAL FAILURE ANALYSIS  *
          *                              *
          *  ITERATION NO.  SCALE FACTOR *
          *  -------------  ------------- *
          *        4          0.151E+03  *
          *                              *
          ********************************

* NUMBER OF ELEMENTS DEGRADED =        6
```

```
              ********************************
              *                              *
              *   MATERIAL FAILURE ANALYSIS  *
              *                              *
              *  ITERATION NO.  SCALE FACTOR *
              *  -------------  ------------ *
              *       5           0.151E+03  *
              *                              *
              ********************************

* NUMBER OF ELEMENTS DEGRADED =       10

              ********************************
              *                              *
              *   MATERIAL FAILURE ANALYSIS  *
              *                              *
              *  ITERATION NO.  SCALE FACTOR *
              *  -------------  ------------ *
              *       6           0.151E+03  *
              *                              *
              ********************************

* NUMBER OF ELEMENTS DEGRADED =       20

              ********************************
              *                              *
              *   MATERIAL FAILURE ANALYSIS  *
              *                              *
              *  ITERATION NO.  SCALE FACTOR *
              *  -------------  ------------ *
              *       7           0.151E+03  *
              *                              *
              ********************************

* NUMBER OF ELEMENTS DEGRADED =       54

              ********************************
              *                              *
              *   MATERIAL FAILURE ANALYSIS  *
              *                              *
              *  ITERATION NO.  SCALE FACTOR *
              *  -------------  ------------ *
              *       8           0.151E+03  *
              *                              *
              ********************************

* NUMBER OF ELEMENTS DEGRADED =      140

              ********************************
              *                              *
              *   MATERIAL FAILURE ANALYSIS  *
              *                              *
              *  ITERATION NO.  SCALE FACTOR *
              *  -------------  ------------ *
```

```
        *        9              0.151E+03    *
        *                                    *
        **************************************

* NUMBER OF ELEMENTS DEGRADED =      188

* ELEMENT FAILURE HAS ALTERED MODEL STABILITY SUCH
  THAT RIGID BODY MODES HAVE BEEN DETECTED.

      TOTAL FAILURE IS ASSUMED.

        ELEMENT PLY FAILURE STATUS AT CYCLE NO     1
```

ELEMENT ID	PLY FAILURE MODES: %FIBER	%MATRIX	%BUCKLING	%TOTAL
5129	0	0	0	100
5130	0	0	0	100
5131	0	0	0	33
5132	0	0	0	33
		.		
		.		
		.		
11461	0	0	0	100
11462	0	0	0	100
11463	0	0	0	100
11464	0	0	0	33

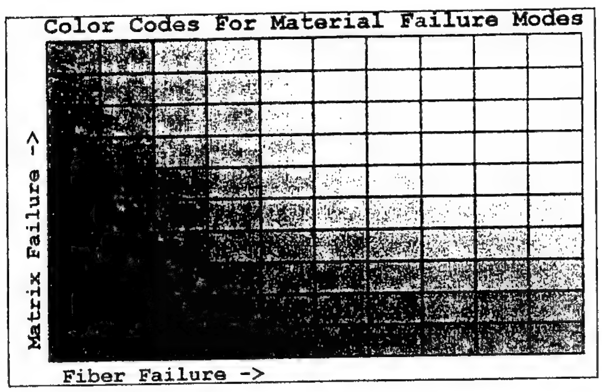
```
################################################
##                                            ##
## ANALYSIS PREDICTS CATASTROPHIC FAILURE OF  ##
## THE MODEL AT AN ULTIMATE LOAD GIVEN BY:    ##
##                                            ##
##    P(ULT) = (0.15140E+03) * P(INITIAL)     ##
##                                            ##
################################################

@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@
@@                                                    @@
@@ F A I L U R E   A N A L Y S I S   C O M P L E T E D @@
@@                                                    @@
@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@
```

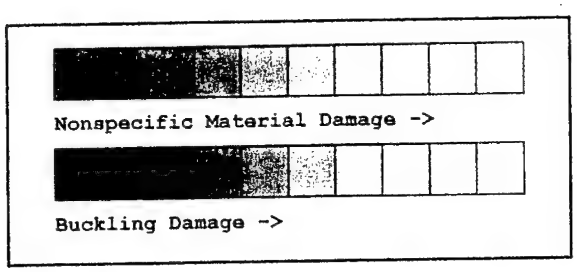
7.3.3 Graphical Output Using MATHEMATICA

The initial graphics generated by MATHEMATICA show the color codes used to display various element failure modes. In the specific demonstration problem analyzed, complete element failure is indicated by a wireframe depiction; because no specific failure modes were predicted using the maximum stress criterion, material failure is shown using the general material failure color coding scheme.

GRAPHICAL DEPICTION OF FAILURE MODES

Fiber and matrix damage are shown using a bichromatic color code. Nonspecific material failure is indicated using a monochrome scheme. Buckling failure is shown in grayscale. Complete failure is indicated by a wireframe depiction.





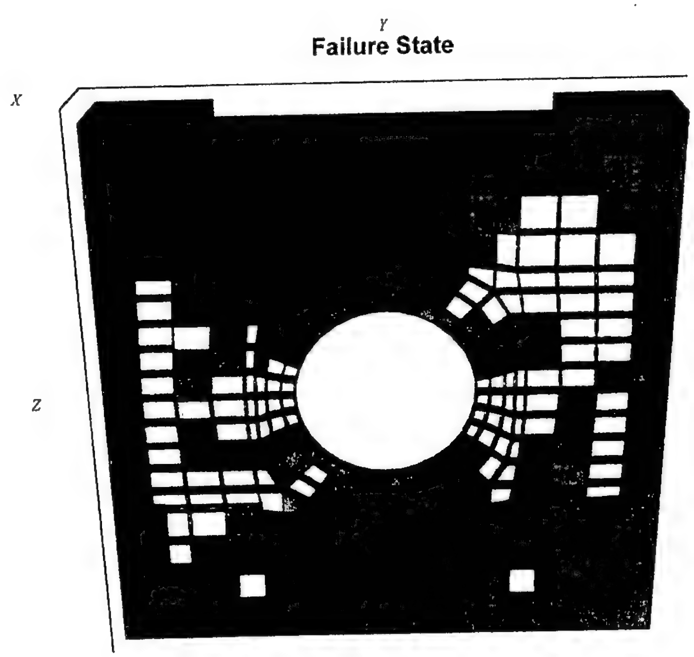
Equilibrium between external and internal forces was converged in six iterations. With the material-only failure option set, in each element all plies were analyzed using the maximum stress criterion to determine the scale factor to by applied to the external loads to cause failure in a specific ply. The lowest factor was selected, multiplied by the acceleration factor (which in this sample run was selected as 1.05 or 5% above the minimum ply failure load), and then applied to the external loads. During the first analysis cycle, the initial failure of three plies was sufficient to cause a sequence of load redistribution and subsequent additional failures that led to a cascade process resulting in ultimate total failure as listed in Table 3.

Table 3. Cascade failure in residual strength analysis.

Iteration	Element Failures
1	3
2	14
3	7
4	6
5	10
6	20
7	54
8	140
9	188

This was automatically determined by the presence of fragmentation which caused the global stiffness matrix to become singular, which terminated execution. The final failure state is shown below. Mediating element failures are shown to have progressed to the outer boundary. Failure in the line of elements comprising the outer boundary have been precluded using the LAPFAIL parameter, which prevents elements to which applied external loads have been applied from exhibiting failure.

```
ANALYSIS CYCLE      1

Material Failure at Scale = 0.151E+03
```



The final residual strength prediction yields the ultimate load carrying capability as a multiple of the initial biaxial loads equal to 151.4. Because a 5% acceleration was used, the resolution of the residual strength measure R falls within the bounds $144.19 \leq R \leq 151.4$.

8 Conclusion

The RESTRAN computer program is a finite element based design tool for the analysis of residual strength in composite structures with arbitrary three-dimensional geometry, loading, support conditions, and material properties. The unique feature of RESTRAN is the combined algorithmic accounting of both material and structural failure modes. Material failure modes are predicted using a robust suite of failure criteria and damage laws. Structural failure due to sublaminate buckling of multiple delaminations is performed, allowing for an arbitrary number of assumed delaminations. User-defined subroutine interfaces are provided to permit user modification and enhancement of basic analysis procedures. A progressive failure analysis is performed until ultimate structural failure is simulated, yielding an estimate of the residual strength. Thus, the RESTRAN program constitutes a robust design/research tool which can provide an accurate assessment of remaining strength in composite structures containing impact damage.

INTENTIONALLY LEFT BLANK.

9 References

1. Wolfram, S. *MATHEMATICA: A System for Doing Mathematics by Computer*, Second Edition, New York, Addison Wesley, 1993.

2. Amtec Engineering, Inc., *TECPLOT, Version 7*, Bellevue, Washington, 1996.

3. Saether, E., "RESTRAN: *RE*sidual *STR*ength *AN*alysis. Vol. I: Theoretical Manual" ARL-TR- , U.S. Army Research Laboratory, Aberdeen Proving Ground, MD, 2001.

4. Hibbitt, Karlsson, and Sorensen, Inc. *ABAQUS User's Manual, Version 5.2*, Pawtucket, RI, 1992.

5. MSC Software Corp., *MSC-NASTRAN, Quick Reference Guide*, Costa Mesa, CA 2001.

6. PDA Engineering, PATRAN Division, *PATRAN Plus User Manual, Release 2.5*, Costa Mesa, CA, 1990.

INTENTIONALLY LEFT BLANK.

Appendix A:

Sample User-Defined Subroutine to Compute Nonlinear Material Properties

INTENTIONALLY LEFT BLANK.

```
      SUBROUTINE USERNL ( STRESS,STRAIN,SXX,SYY,SZZ,TYZ,TZX,TXY,EXX,
     1                    EYY,EZZ,GYZ,GZX,GXY,THKN,ATHK,TLM,THETA,
     2                    PNLRM,FE1,FE2,FE3,FG31,FG23,FG12,FV31,
     3                    FV23,FV12,NORD )
C
C     ***********************************************
C     **                                           **
C     **   USER-DEFINED ROUTINE TO COMPUTE SECANT  **
C     **   MODULI IN NONLINEAR MATERIAL ANALYSIS   **
C     **                                           **
C     ***********************************************
C
      IMPLICIT DOUBLE PRECISION(A-H,O-Z)
C
      DIMENSION STRESS(27,6),STRAIN(27,6),PNLRM(25)
C
C @@@ THREE DIFFERENT USER-DEFINED NONLINEARITIES  ARE  CONTAINED @@@
C @@@ IN THIS SUBROUTINE. THE FIRST SPECIFIES A PIECE-WISE LINEAR @@@
C @@@ LINEAR  STRESS-STRAIN  RELATIONSHIP,  THE  SECOND   DEFINES @@@
C @@@ UNEQUAL  MODULI IN TENSION AND COMPRESSION. THE THIRD  CASE @@@
C @@@ YIELDS MODULI THAT ARE A CONTINUOUS  FUNCTION  OF  STRESSES @@@
C @@@ UP TO SET LEVEL. THESE CASES ARE SELECTED  THROUGH  SETTING @@@
C @@@ THE FIRST ELEMENT OF THE  PNLRM  ARRAY  WHICH  CONTAINS, IN @@@
C @@@ SEQUENCE, THE PARAMETERS, Pn, SPECIFIED IN  THE  *NONLINEAR @@@
C @@@ MATERIAL INPUT STATEMENT.                                   @@@
C
      IF ( INT(PNLRM(1)) .EQ. 1 ) THEN
C
C    @@@ USER-DEFINED PIECE-WISE LINEAR STRESS-STRAIN LAW @@@
C    @@@                                                  @@@
C    @@@ PNLRM(2,3,4) : CONTAINS YIELD STRESS VALUES      @@@
C    @@@ PNLRM(5,6,7) : CONTAINS MODULI REDUCTION FACTOR  @@@
C
C    @@@ COMPUTE VON MISES STRESS @@@
C
         VMSS = SQRT(( (SXX-SYY)**2 + (SYY-SZZ)**2 + (SZZ-SXX)**2)/2.0)
C
C    @@@ DETERMINE WHICH LINEAR SECTION OF THE @@@
C    @@@ STRESS/STRAIN CURVE WE ARE IN         @@@
C
         IF ( VMSS .LT. PNLRM(1) ) THEN
C
C      @@@ INITIAL MODULI @@@
C
            FAC = 1
C
         ELSE IF ( VMSS .GE. PNLRM(1) .AND. VMSS .LT. PNLRM(2) ) THEN
C
C      @@@ SECOND LINEAR SEGMENT @@@
C
            FAC = PNLRM(4)
C
         ELSE IF ( VMSS .GE. PNLRM(2) .AND. VMSS .LT. PNLRM(3) ) THEN
C
```

```
C       @@@ THIRD LINEAR SEGMENT @@@
C
            FAC = PNLRM(5)
C
         ELSE IF ( VMSS .GE. PNLRM(3) ) THEN
C
C       @@@ FINAL LINEAR SEGMENT @@@
C
            FAC = PNLRM(5)
C
         END IF
C
C    @@@ ASSIGN MODULI REDUCTION FACTORS @@@
C
         FE1  = FAC
         FE2  = FAC
         FE3  = FAC
         FG31 = FAC
         FG23 = FAC
         FG12 = FAC
C
      ELSE IF ( INT(PNRLM(1)) .EQ. 2 ) THEN
C
C    @@@       UNEQUAL TENSION AND COMPRESSION MODULI            @@@
C    @@@                                                         @@@
C    @@@ PNRLM(2-7)  CONTAIN MULTIPLIERS TO YIELD TENSION MODULI @@@
C    @@@             VALUES FOR Exx,Eyy,Ezz,Gzx,Gyz,Gxy          @@@
C    @@@ PNRLM(8-15) CONTAIN MULTIPLIERS TO YIELD COMPRESSION    @@@
C    @@@             MODULI VALUES FOR Exx,Eyy,Ezz,Gzx,Gyz,Gxy   @@@
C
         IF ( SXX .GT. 0.0D0 ) THEN
            FE1 = PNRLM(2)
         ELSE
            FE1 = PNRLM(8)
         END IF
         IF ( SYY .GT. 0.0D0 ) THEN
            FE2 = PNRLM(3)
         ELSE
            FE2 = PNRLM(9)
         END IF
         IF ( SZZ .GT. 0.0D0 ) THEN
            FE3 = PNRLM(4)
         ELSE
            FE3 = PNRLM(10)
         END IF
         IF ( TYZ .GT. 0.0D0 ) THEN
            FG31 = PNRLM(5)
         ELSE
            FG31 = PNRLM(11)
         END IF
         IF ( TZX .GT. 0.0D0 ) THEN
            FG23 = PNRLM(6)
         ELSE
            FG32 = PNRLM(12)
```

```
            END IF
            IF ( TXY .GT. 0.0D0 ) THEN
               FG12 = PNRLM(7)
            ELSE
               FG12 = PNRLM(13)
            END IF
C
          ELSE IF ( INT(PNRLM(1)) .EQ. 3 ) THEN
C
C     @@@ USER-DEFINED NONLINEAR STRESS-STRAIN LAW    @@@
C     @@@                                             @@@
C     @@@ PNRLM(2-11)  : CONTAINS ULTIMATE STRENGTHS  @@@
C     @@@                   SXX-TEN, SXX-COMP,        @@@
C     @@@                   SYY-TEN, SYY-COMP,        @@@
C     @@@                   SZZ-TEN, SZZ-COMP,        @@@
C     @@@                   R, S, T                   @@@
C
C     @@@ REDUCTION LAW: FACi = 1.0 - BETA*(Si/STRENGTHi) @@@
C
            BETA1 = 0.75
            BETA2 = 0.50
            IF ( SXX .GE. 0.0 ) THEN
               FE1 = 1.0 - BETA1*ABS(SXX/PNLRM(2))
            ELSE
               FE1 = 1.0 - BETA1*ABS(SXX/PNLRM(3))
            END IF
C
            IF ( SYY .GE. 0.0 ) THEN
               FE2 = 1.0 - BETA1*ABS(SYY/PNLRM(4))
            ELSE
               FE2 = 1.0 - BETA1*ABS(SYY/PNLRM(5))
            END IF
C
            IF ( SZZ .GE. 0.0 ) THEN
               FE3 = 1.0 - BETA1*ABS(SZZ/PNLRM(6))
            ELSE
               FE3 = 1.0 - BETA1*ABS(SZZ/PNLRM(7))
            END IF
C
            FG31 = 1.0 - BETA2*ABS(TYZ/PNLRM(8))
            FG23 = 1.0 - BETA2*ABS(TZX/PNLRM(9))
            FG12 = 1.0 - BETA2*ABS(TXY/PNLRM(10))
C
C     @@@ LIMIT MODULI REDUCTION TO FACTOR OBTAINED TO 25% @@@
C     @@@ FOR YOUNG'S MODULI AND 50% FOR SHEAR MODULI      @@@
C
            IF ( FE1  .LT. 0.75 ) FE1  = 0.75
            IF ( FE2  .LT. 0.75 ) FE2  = 0.75
            IF ( FE3  .LT. 0.75 ) FE3  = 0.75
            IF ( FG31 .LT. 0.50 ) FG31 = 0.50
            IF ( FG23 .LT. 0.50 ) FG23 = 0.50
            IF ( FG12 .LT. 0.50 ) FG12 = 0.50
C
          END IF
```

```
C
C @@@ END EXAMPLE @@@
C
      RETURN
      END
```

Appendix B:

Sample User-Defined Subroutine to Compute Material Failure

INTENTIONALLY LEFT BLANK.

```
      SUBROUTINE USERFC ( SGPTS,EGPTS,STRESS,STRAIN,PTHK,THETA,IPLY,
     1                    NELID,PARAM,E1,E2,E3,G12,G23,G31,V12,V23,
     2                    V13,EM,EF,NORD,NSTAT,NFSTAT,NACCS )
C
C     *************************************************
C     **                                             **
C     **  USER-DEFINED FAILURE CRITERION TO COMPUTE  **
C     **  MATERIAL FAILURE UNDER APPLIED LOAD        **
C     **                                             **
C     *************************************************
C
      IMPLICIT DOUBLE PRECISION(A-H,O-Z)
C
      DIMENSION SGPTS(27,6),EGTPS(27,6),STRESS(6),STRAIN(6)
      DIMENSION PARAM(15)
C
C     ### BEGIN EXAMPLE ###
C
C <<< VALUE IN PARAM(1) IS USED TO SELECT THE >>>
C <<< FAILURE CRITERION TO BE APPLIED. OTHER  >>>
C <<< VALUES STORED IN PARAM ARRAY AS NEEDED. >>>
C
      NFLAG = INT(PARAM(1))
C
      IF ( NFLAG .EQ. 1 ) THEN
C
C    <<< FAILURE CRITERIA AND DAMAGE LAWS AS PRESENTED IN:     >>>
C    <<< FU-KUO CHANG AND KUO-YEN CHANG, "A PROGRESSIVE DAMAGE >>>
C    <<< MODEL FOR LAMINATED COMPOSITES CONTAINING STRESS      >>>
C    <<< CONCENTRATION," J. COMP. MAT, PP. 834-855, 1987.      >>>
C
        S1  = STRESS(1)
        S2  = STRESS(2)
        S12 = STRESS(6)
C
C    <<< NONLINEAR SHEAR STRESS-STRAIN FACTOR ALPHA >>>
C
        ALPHA = PARAM(2)
C
C    <<< ULTIMATE STRENGTH MEASURES >>>
C
        XT = PARAM(3)
        YT = PARAM(4)
        SC = PARAM(5)
C
        EM = SQRT( (S2/YT)**2 + (S12**2/(2*G12)+3*ALPHA*S12**4/4)/
     1                          (SC**2 /(2*G12)+3*ALPHA* SC**4/4) )
C
C    <<< FIBER-MATRIX SHEARING AND FIBER BREAKAGE >>>
C
        EF = SQRT( (S1/XT)**2 + (S12**2/(2*G12)+3*ALPHA*S12**4/4)/
     1                          (SC**2 /(2*G12)+3*ALPHA* SC**4/4) )
C
        ET = MAX(EM,EF)
```

```
      EF = 1.0/ET
      EM = EF
C
      NFSTAT = NSTAT
      NACCS  = 0
C
C   <<< EMPLOY PROPERTY DEGRADATION LAWS >>>
C
      IF ( EM .GT. 1.0 ) THEN
C
C      <<< MATRIX CRACKING >>>
C
         V12 = 0.0
         V23 = 0.0
         V13 = 0.0
         E2  = 0.0
         E3  = 0.0
         G23 = 0.0
C
         IF ( NSTAT .EQ. 0 ) THEN
           NFSTAT = 2
         ELSE
           NFSTAT = 3
         END IF
         NACCS = 1
C
       END IF
C
       IF ( EF .GT. 1.0 ) THEN
C
C       <<< FIBER BREAKAGE/FIBER-MATRIX SHEARING DAMAGE CALCULATED >>>
C       <<< USING A MICROMECHANICAL FIBER BUNDLE FAILURE APPROACH  >>>
C
         A    = PTHK
         AO   = PARAM(6)
         BETA = PARAM(7)
C
C      <<< DAMAGE IS ASSUMED TO FOLLOW A WEIBUL DISTRIBUTION >>>
C
         E1  = E1* EXP(-(A/AO)**BETA)
         G12 = G12*EXP(-(A/AO)**BETA)
         G31 = G31*EXP(-(A/AO)**BETA)
C
         IF ( NSTAT .EQ. 0 ) THEN
           NFSTAT = 2
         ELSE
           NFSTAT = 3
         END IF
         NACCS = 1
C
       END IF
C
    END IF
C
```

```
C *** END EXAMPLE ***
C
      RETURN
      END
```

INTENTIONALLY LEFT BLANK.

Appendix C:

Sample User-Defined Subroutine to Assign Initial Material Damage

INTENTIONALLY LEFT BLANK.

```
      SUBROUTINE USERID (PTHK,THETA,E1,E2,E3,G13,G23,G12,V13,V23,
     1                   V12,P1,P2,P3,P4,P5,P6,P7,P8,P9,NELID,
     2                   IPLY,NSTAT )
C
C     *************************************************
C     **                                             **
C     **     USER-DEFINED SUBROUTINE TO ASSIGN       **
C     **     INITIAL ELEMENT MATERIAL DAMAGE.        **
C     **                                             **
C     *************************************************
C
      IMPLICIT DOUBLE PRECISION(A-H,O-Z)
C
C             *** EXAMPLE ROUTINE ***
C
C @@@ USER DEFINITION FOR THE INITIAL PROPERTIES OF       @@@
C @@@ Kth PLY IN ELEMENT WITH ID LABEL NELID:             @@@
C
C @@@ SELECTION OF INITIAL DAMAGE IS BASED ON ELEMENT     @@@
C @@@ ID SERIES AND SPECIALIZED MATERIAL PROPERTY         @@@
C @@@ DEGRADATION RELATIONS USING INPUT PARAMETERS        @@@
C @@@ P1 -> P9                                            @@@
C
      IF ( NELID .GE. 2000 .AND. NELID .LT. 3000 ) THEN
C
C    @@@ PARTIAL FAILURE I. TRANSVERSE MODULI REDUCTION    @@@
C
        E2  = P1*E2
        G12 = P1*G12
        G23 = P2*G23
C
        NSTAT = 2
C
      ELSE IF ( NELID .GE. 3000 .AND. NELID .LT. 4000 ) THEN
C
C    @@@ PARTIAL FAILURE II. TRANSVERSE INPLANE AND NORMAL @@@
C    @@@ MODULI REDUCTION                                  @@@
C
        E2  = P3*E2
        E3  = P5*E3
        G12 = P3*G12
        G23 = P4*G23
        G13 = P6*G13
C
        NSTAT = 2
C
      ELSE IF ( NELID .GE. 4000 .AND. NELID .LT. 5000 ) THEN
C
C    @@@ PARTIAL FAILURE III. TRANSVERSE INPLANE AND       @@@
C    @@@ NORMAL MODULI REDUCTION                           @@@
C
        E2  = P3*E2
        E3  = P5*E3
        G12 = P3*G12
```

```
        G23 = P4*G23
        G13 = P6*G13
C
        NSTAT = 2
C
      ELSE IF ( NELID .GE. 5000 .AND. NELID .LT. 7000 ) THEN
C
C    @@@ TOTAL FAILURE. REDUCE ALL MATERIAL PROPERTIES     @@@
C    @@@ TO ZERO                                           @@@
C
        E1  = 0.0
        E2  = 0.0
        E3  = 0.0
        G13 = 0.0
        G23 = 0.0
        G12 = 0.0
        V13 = 0.0
        V23 = 0.0
        V12 = 0.0
C
        NSTAT = 3
C
      END IF
C
C                 *** END EXAMPLE ***
C
      RETURN
      END
```

Appendix D:

Sample User-Defined Subroutine to Interpret Buckling Modes

INTENTIONALLY LEFT BLANK.

```
      SUBROUTINE USERB1 ( EMODE,NELEM,NDSET,NXSET,NSETI,NDID,EIGV,
     1                    NUMNOD,NBPRC,LCNSL,ITERB,NUMEXC, NELNUM,
     2                    NACC,NSURF,NINC,MDX0,MDX1,MAXB,MDX3 )
C
C     ************************************************
C     **                                            **
C     **     USER-DEFINED SUBROUTINE TO EVALUATE    **
C     **     BUCKLING FAILURE MODE                  **
C     **                                            **
C     ************************************************
C
      IMPLICIT DOUBLE PRECISION(A-H,O-Z)
C
      DIMENSION EMODE(MDX0,MAXB),NELEM(MDX3,12),NDSET(MDX1)
      DIMENSION NXSET(MDX1)
C
C     ### BEGIN EXAMPLE ###
C
C @@@ DESCRIPTION: FOR DELAMINATION NODE SETS 10 AND 20, ELEMENT   @@@
C @@@ PAIRS HAVE BEEN IDENTIFIED ON EITHER DELAMINATION FACE WHICH @@@
C @@@ WILL INDICTATE BUCKLING IF THEY SHOW THE SAME SIGN OF         @@@
C @@@ RELATIVE MOTION AND IF THE RELATIVE DEFORMATION IS GREATER   @@@
C @@@ THAN 0.4. THIS IS BASED ON THE FACT THAT THE MODE SHAPE IS   @@@
C @@@ NORMALIZED AND THAT A PARTICIPATION OF THE SELECTED ELEMENTS @@@
C @@@ GREATER THAN 40% OF THE MAXIMUM DEFLECTION IS TAKEN AS A     @@@
C @@@ SIGNIFICANT PARTICIPATION IN THE MODE. THE SIGNS OF THE      @@@
C @@@ DISPLACEMENTS INDICATING AN OPENING OR CLOSING MODE ARE      @@@
C @@@ ARBITRARY SINCE THE MAGNITUDE OF THE MODE SHAPE IS           @@@
C @@@ INDETERMINATE.                                               @@@
C
      NACCS = 0
C
      IF ( NDID .EQ. 10 ) THEN
C
C    @@@ DELAMINATION ID 10 IS BEING PROCESSED. BUCKLING WILL BE @@@
C    @@@ BASED ON THE MAGNITUDE OF RELATIVE DISPLACEMENT BETWEEN @@@
C    @@@ OPPOSING ELEMENTS 2050 AND 3050 WHICH ARE LOCATED ABOVE @@@
C    @@@ AND BELOW THE DELONANE.                         @@@
C
C    @@@ OBTAIN THE FOLLOWING DISPLACEMENTS: @@@
C    @@@       NODE 1 IN ELEMENT 2050       @@@
C    @@@       NODE 5 IN ELEMENT 3050       @@@
C
C    @@@ LOCATE ELEMENTS IN ARRALEEQ. 2050 ) NL2050 = I
          IF ( NELEM(I,1) .EQ. 3050 ) NL3050 = I
        END DO
C
C   @@@ CONVERT INTERNAL NODE NUMBERS IN NELEM ARRAY TO @@@
C   @@@ INPUT NODE NUMBERS                              @@@
C
        NODE1 = NSETI(NELEM(NL2050,2))
        NODE5 = NSETI(NELEM(NL3050,6))
C
C   @@@ LOCATE NODE MODAL DEFLECTIONS IN EMODE ARRAY. TEST @@@
```

```
C    @@@ ON INPUT NODE NUMBER AND GENERATED COINCIDENT NODE @@@
C    @@@ NUMBER STORED IN SAME RECORD                       @@@
C
        NPNT1 = 0
        NPNT5 = 0
        DO I = 1, NUMNOD
          IF ( NODE1 .EQ. INT(EMODE(I,1)) .OR.
     1         NODE1 .EQ. INT(EMODE(I,5)) ) NPNT1 = I
          IF ( NODE5 .EQ. INT(EMODE(I,1)) .OR.
     1         NODE5 .EQ. INT(EMODE(I,5)) ) NPNT5 = I
        END DO
C
C    @@@ OBTAIN W-COMPONENT FOR COMPARISON @@@
C
        IF ( NODE1 .EQ. INT(EMODE(NPNT1,1)) ) W1 = EMODE(NPNT1,4)
        IF ( NODE1 .EQ. INT(EMODE(NPNT1,5)) ) W1 = EMODE(NPNT1,8)
        IF ( NODE5 .EQ. INT(EMODE(NPNT5,1)) ) W5 = EMODE(NPNT5,4)
        IF ( NODE5 .EQ. INT(EMODE(NPNT5,5)) ) W5 = EMODE(NPNT5,8)
C
        IF ( ABS(W1-W5) .GT. 0.40 ) THEN
C
C      @@@ LOCAL BUCKLING IS DETERMINED @@@
C
          NACC  = 1
          NSURF = 1
C
        ELSE
C
          NACC = 0
C
        END IF
C
      END IF
C
      RETURN
      END
```

Appendix E:

Sample User-Defined Subroutine to Assign Material Degradation Due to Buckling

INTENTIONALLY LEFT BLANK.

```
      SUBROUTINE USERB2 ( NELEM,NODSET,NDELM,NDID,NELFL,ELFAC,NFAIL,
     1                    NUMELF,NSURF,MDX1,MDX2,MDX3,MDX4 )
C
C     ***********************************************
C     **                                           **
C     **    USER-DEFINED SUBROUTINE TO DETERMINE   **
C     **    ELEMENTS FAILED DUE TO BUCKLING  AND   **
C     **    ASSIGNED REDUCED MATERIAL PROPERTIES   **
C     **                                           **
C     ***********************************************
C
      IMPLICIT DOUBLE PRECISION(A-H,O-Z)
C
      DIMENSION NELEM(MDX3,12),NODSET(MDX2)
      DIMENSION NELFL(MDX1),ELFAC(MDX1),NDELM(MDX4)
C
      NUMELF   = 0
      NELFL(1) = 0
      ELFAC(1) = 0.0
C
C     ### BEGIN EXAMPLE ###
C
C     @@@ ASSIGN DIFFERENT LEVELS OF MATERIAL FAILURE DUE @@@
C     @@@ BUCKLING DEPENDING ON THE DELAMINATION LOCATION @@@
C     @@@ WHICH IS SIGNIFIED BY THE DELAMINATION ID       @@@
C
      IF ( NDID .EQ. 10 ) THEN
C
C   @@@ PARTIAL FAILURE @@@
C
C   @@@ REDUCE MODULI BY 50% IN ELEMENTS 5000 THRU 5500. @@@
C   @@@ NUMBER OF FAILED ELEMENTS RETURNED IN NUMELF     @@@
C
        NUMELF  = 501
        DO I = 1, 501
          NELFL(I) = 4999+I
          ELFAC(1) = 0.50
        END DO
C
      END IF
C
      RETURN
      END
```

INTENTIONALLY LEFT BLANK.

Appendix F:

Sample Generated MATHEMATICA Input File

INTENTIONALLY LEFT BLANK.

```
(*
                    @@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@
                    @@                             @@
                    @@    MATHEMATICA INPUT FILE   @@
                    @@                             @@
                    @@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@
*)
                  fcode = {
{RGBColor[ 1.0, 1.0, 1.0],Rectangle[{ 0.0,0.0},{10.0, 6.0}]},
{RGBColor[ 0.0, 0.5, 1.0],Rectangle[{ 0.0,3.0},{10.0, 4.0}]},
 Text["RESTRAN GRAPHICAL RESULTS",{5.0,3.5}]
                         }
                  Show[Graphics[ fcode ] ]

                  model1 = {
{RGBColor[ 1.0, 1.0, 1.0],Rectangle[{ 0.0,0.5},{10.0, 5.0}]},
{Text[FontForm["ANALYSIS CYCLE     1 ",
{"Courier-Bold",8}],{5.0,4.0},{0,-1}]},
{Text[FontForm["Linear Buckling Analysis",
{"Courier-Bold",8}],{5.0,3.5}]},
 Line[{{0.0,2.75},{10.0,2.75},{10.0,4.5},{0.0,4.5},{0.0,2.75}}]
                           }
                   Show[Graphics[model1,
                        Frame -> False ] ]
(*
                @@@ UNDEFORMED MODEL GEOMETRY @@@
                @@@ SHOWING MATERIAL FAILURE  @@@
*)
   ppnts1 = {{ 0.0000, 0.0000, 0.9000},{ 2.0000, 0.0000, 0.9000},
             { 2.0000, 0.0000, 1.0000},{ 0.0000, 0.0000, 1.0000}}
   ppnts2 = {{ 0.0000, 0.0000, 0.9000},{ 0.0000, 0.0000, 1.0000},
             { 0.0000, 1.0000, 1.0000},{ 0.0000, 1.0000, 0.9000}}
   ppnts3 = {{ 0.0000, 0.0000, 0.9000},{ 0.0000, 1.0000, 0.9000},
             { 2.0000, 1.0000, 0.9000},{ 2.0000, 0.0000, 0.9000}}
   ppnts4 = {{ 0.0000, 0.0000, 1.0000},{ 2.0000, 0.0000, 1.0000},
             { 2.0000, 1.0000, 1.0000},{ 0.0000, 1.0000, 1.0000}}
                                 .
                                 .
                                 .
                  model2      = {
{FaceForm[RGBColor[.00,.00,.99],RGBColor[.00,.00,.99]],Polygon[ppnts1]},
{FaceForm[RGBColor[.00,.00,.99],RGBColor[.00,.00,.99]],Polygon[ppnts2]},
{FaceForm[RGBColor[.00,.00,.99],RGBColor[.00,.00,.99]],Polygon[ppnts3]},
                                 .
                                 .
                                 .
                                  }

          Show[ Graphics3D[model2,
                Boxed -> False,
                Axes -> True,
                Ticks -> None,
                Lighting -> False,
                RenderAll -> True,
```

```
              ViewPoint -> {1.3,-2.4,2.0},
              ViewVertical -> {0,0,1},
              DefaultFont -> {"Times-Italic",6},
              PlotLabel -> FontForm["Initial Geometry",
                         {"Helvetica-Bold",9}],
              AxesLabel -> {"X","Y","Z"} ] ]

     ClearAll[ppnts1,ppnts2,ppnts3,ppnts4,ppnts5]
(*
              @@@ DEFORMED MODEL GEOMETRY @@@

     THE FOLLOWING AMPLITUDE FACTOR, GAMMAi, CAN BE
     CHANGED TO GIVE THE DESIRED DEFORMED SHAPE
*)
                gamma1       =     -1.5000

  tpnts1 = {0.0000,0.0000,0.9000} + gamma1{0.0000,0.0000,-1.0000}
  tpnts2 = {2.0000,0.0000,0.9000} + gamma1{0.0188,0.0000,-0.6335}
  tpnts3 = {2.0000,0.0000,1.0000} + gamma1{-.0182,0.0000,-0.6335}
  tpnts4 = {0.0000,0.0000,1.0000} + gamma1{0.0000,0.0000,-1.0000}
  ppnts223    = {tpnts1     ,tpnts2     ,tpnts3     ,tpnts4     }
(*
              @@@ DEFORMED MODEL GEOMETRY @@@
*)
       model3      =              {
  Polygon[ppnts223 ],Polygon[ppnts224 ],Polygon[ppnts225 ],
  Polygon[ppnts226 ],Polygon[ppnts227 ],Polygon[ppnts228 ],
                      .
                      .
                      .
                                  }
       Show[ Graphics3D[model3,
             Boxed -> False,
             Axes -> True,
             Ticks -> None,
             Lighting -> True,
             RenderAll -> True,
             ViewPoint -> {1.3,-2.4,2.0},
             ViewVertical -> {0,0,1},
             DefaultFont -> {"Times-Italic",6},
             PlotLabel ->
             FontForm["Buckling Mode",
                    {"Helvetica-Bold",9}],
             AxesLabel -> {"X","Y","Z"} ]  ]

  ClearAll[tpnts1  ,tpnts2  ,tpnts3  ,tpnts4  ,tpnts5  ]
  ClearAll[tpnts6  ,tpnts7  ,tpnts8  ,tpnts9  ,tpnts10 ]
```

Appendix G

Sample User-Defined Subroutine to Output Model Data for Graphical Display

INTENTIONALLY LEFT BLANK.

```
      SUBROUTINE USRGRF (NELEM,COORDS,U,NSETI,NELNUM,NUMNOD,
     1                   NWTRN,NDFRM,GMFAC,SCALE,NPATH,NTG,
     2                   NFLAG,NDEV,NTOTF,MDX1,MDX2,MDX3 )
C
C     ******************************************
C     **                                      **
C     **  USER-DEFINED GRAPHICS OUTPUT ROUTINE  **
C     **                                      **
C     ******************************************
C
      IMPLICIT DOUBLE PRECISION(A-H,O-Z)
C
      DIMENSION NELEM(MDX3,12),COORDS(MDX2,4),U(MDX1)
      DIMENSION NSETI(MDX1)
C
C
C     NODE CONVENTION
C
C                        Y,V
C             Z,W         /
C             |          /            SURFACE
C             |   8---------7      FACE   NODES
C             |  /|        /|       F1: (1,2,6,5)
C             | / |       / |       F2: (2,3,7,6)
C             |/  |      /  |       F3: (3,4,8,7)
C             5---------6   |       F4: (1,5,8,4)
C             |   |     |   |       F5: (1,4,3,2)
C             |   4-----|---3       F6: (5,6,7,8)
C             |  /      |  /
C             | /       | /
C             |/        |/
C             1---------2 --------- X,U
C
C
C <<< PROGRAM STATEMENTS TO OUTPUT GRAPHICAL    >>>
C <<< INFORMATION IN A USER-DEFINED FORMAT      >>>
C <<< ELEMENT FAILURE MEASURES:                 >>>
C <<<                                           >>>
C <<< PR1: PERCENT OF PLIES WITH FIBER FAILURE  >>>
C <<< PR2: PERCENT OF PLIES WITH MATRIX FAILURE >>>
C <<< PR3: PERCENT MEASURE OF BUCKLING FAILURE  >>>
C <<< PR4: PERCENT MEASURE OF TOTAL FAILURE     >>>
C
C <<< NDEV = 15 FOR UNIT ASSOCIATED WITH FILE restran.prp >>>
C <<<          CONTAINING GRAPHICS GENERATED DURING PREPASS >>>
C <<<        = 24 FOR UNIT ASSOCIATED WITH FILE restran.grf >>>
C <<<          FOR GRAPHICS GENERATED DURING ANALYSIS       >>>
C
C <<< IF TOTAL FAILURE FLAG IS SET (NTOTF = 1), THE >>>
C <<< DEFLECTION OR MODE SHAPE VECTOR WILL          >>>
C <<< GENERALLY BE NULL                             >>>
C
C @@@ SAMPLE ROUTINE FOR PROCESSING ELEMENT GEOMETRY @@@
C @@@ AND NODE DISPLACEMENTS FOR DEFORMATION PLOT    @@@
```

```
C
C @@@ DETERMINE COORDINATES OF ELEMENT NODES FOR A    @@@
C @@@ PLOT OF UNDEFORMED GEOMETRY                     @@@
C
      DO I = 1, NELNUM
C
        NELID = NELEM(I,1)
        DO J = 1, 9
          M = NELEM(I,J+1)
C
C     @@@ SEARCH COORDINATE ARRAY FOR ELEMENT NODE @@@
C
          DO K = 1, NUMNOD
            IF ( M .EQ. INT(COORDS(K,1)) ) THEN
              N = K
              GO TO 100
            END IF
          END DO
100       CONTINUE
C
C     @@@ COORDINATES OF NODE M IN ELEMENT I ARE: @@@
C
          CX = COORDS(N,2)
          CY = COORDS(N,3)
          CZ = COORDS(N,4)
C
        END DO
C
      END DO
C
C @@@ EXTRACT DISPLACEMENTS AT NODES TO ADD TO NODE @@@
C @@@ COORDINATES FOR A PLOT OF DEFORMED GEOMETRY   @@@
C
      DO I = 1, NELNUM
C
        DO K = 1, 8
          N = NELEM(I,K+1)
          DO IC = 1, NUMNOD
            IF ( N .EQ. INT(COORDS(IC,1)) ) THEN
               NL = IC
               GO TO 200
            END IF
          END DO
200       CONTINUE
C
C     @@@ COORDINATES FOR THE K+1 NODE IN ELEMENT I DETERMINED @@@
C
          CX = COORDS(NL,2)
          CY = COORDS(NL,3)
          CZ = COORDS(NL,4)
C
C     @@@ DISPLACEMENTS AT THE CURRENT NODE ARE OBTAINED AS @@@
C
          DX = U(3*N-2)
```

```
          DY = U(3*N-1)
          DZ = U(3*N  )
C
C     @@@ THE LOCATION OF THE DISPLACED NODE IS GIVEN BY @@@
C
          DNX = CX + DX
          DNY = CY + DY
          DNZ = CZ + DZ
C
C     @@@ THESE COORDINATES ARE THEN USED TO DEVELOP THE @@@
C     @@@ THE GEOMETRY IN A USER-DEFINED FORMAT TO PLOT  @@@
C     @@@ THE DEFORMED GEOMETRY OF THE MODEL             @@@
C
        END DO
C
      END DO
C
      RETURN
      END
```

INTENTIONALLY LEFT BLANK.

NO. OF COPIES	ORGANIZATION
2	DEFENSE TECHNICAL INFORMATION CENTER DTIC OCA 8725 JOHN J KINGMAN RD STE 0944 FT BELVOIR VA 22060-6218
1	HQDA DAMO FDT 400 ARMY PENTAGON WASHINGTON DC 20310-0460
1	OSD OUSD(A&T)/ODDR&E(R) DR R J TREW 3800 DEFENSE PENTAGON WASHINGTON DC 20301-3800
1	COMMANDING GENERAL US ARMY MATERIEL CMD AMCRDA TF 5001 EISENHOWER AVE ALEXANDRIA VA 22333-0001
1	INST FOR ADVNCD TCHNLGY THE UNIV OF TEXAS AT AUSTIN 3925 W BRAKER LN STE 400 AUSTIN TX 78759-5316
1	DARPA SPECIAL PROJECTS OFFICE J CARLINI 3701 N FAIRFAX DR ARLINGTON VA 22203-1714
1	US MILITARY ACADEMY MATH SCI CTR EXCELLENCE MADN MATH MAJ HUBER THAYER HALL WEST POINT NY 10996-1786
1	DIRECTOR US ARMY RESEARCH LAB AMSRL D DR D SMITH 2800 POWDER MILL RD ADELPHI MD 20783-1197

NO. OF COPIES	ORGANIZATION
1	DIRECTOR US ARMY RESEARCH LAB AMSRL CI AI R 2800 POWDER MILL RD ADELPHI MD 20783-1197
3	DIRECTOR US ARMY RESEARCH LAB AMSRL CI LL 2800 POWDER MILL RD ADELPHI MD 20783-1197
3	DIRECTOR US ARMY RESEARCH LAB AMSRL CI IS T 2800 POWDER MILL RD ADELPHI MD 20783-1197
	ABERDEEN PROVING GROUND
2	DIR USARL AMSRL CI LP (BLDG 305)

NO. OF COPIES	ORGANIZATION
1	DIRECTOR US ARMY RESEARCH LAB AMSRL CP CA D SNIDER 2800 POWDER MILL RD ADELPHI MD 20783-1145
1	DIRECTOR US ARMY RESEARCH LAB AMSRL OP SD TA 2800 POWDER MILL RD ADELPHI MD 20783-1145
3	DIRECTOR US ARMY RESEARCH LAB AMSRL OP SD TL 2800 POWDER MILL RD ADELPHI MD 20783-1145
1	DIRECTOR US ARMY RESEARCH LAB AMSRL CI IS T 2800 POWDER MILL RD ADELPHI MD 20783-1145
1	DIRECTOR DA OASARDA SARD SO 103 ARMY PENTAGON WASHINGTON DC 20310-0103
1	DPTY ASST SECY FOR R&T SARD TT THE PENTAGON RM 3EA79 WASHINGTON DC 20301-7100
1	COMMANDER US ARMY MATERIEL CMD AMXMI INT 5001 EISENHOWER AVE ALEXANDRIA VA 22333-0001
4	COMMANDER US ARMY ARDEC AMSTA AR CC G PAYNE J GEHBAUER C BAULIEU H OPAT PICATINNY ARSENAL NJ 07806-5000

NO. OF COPIES	ORGANIZATION
2	COMMANDER US ARMY ARDEC AMSTA AR AE WW E BAKER J PEARSON PICATINNY ARSENAL NJ 07806-5000
1	COMMANDER US ARMY ARDEC AMSTA AR TD C SPINELLI PICATINNY ARSENAL NJ 07806-5000
1	COMMANDER US ARMY ARDEC AMSTA AR FSE PICATINNY ARSENAL NJ 07806-5000
6	COMMANDER US ARMY ARDEC AMSTA AR CCH A W ANDREWS S MUSALLI R CARR M LUCIANO E LOGSDEN T LOUZEIRO PICATINNY ARSENAL NJ 07806-5000
1	COMMANDER US ARMY ARDEC AMSTA AR CCH P J LUTZ PICATINNY ARSENAL NJ 07806-5000
1	COMMANDER US ARMY ARDEC AMSTA AR FSF T C LIVECCHIA PICATINNY ARSENAL NJ 07806-5000
1	COMMANDER US ARMY ARDEC AMSTA ASF PICATINNY ARSENAL NJ 07806-5000

NO. OF COPIES	ORGANIZATION
1	COMMANDER US ARMY ARDEC AMSTA AR QAC T C C PATEL PICATINNY ARSENAL NJ 07806-5000
1	COMMANDER US ARMY ARDEC AMSTA AR M D DEMELLA PICATINNY ARSENAL NJ 07806-5000
3	COMMANDER US ARMY ARDEC AMSTA AR FSA A WARNASH B MACHAK M CHIEFA PICATINNY ARSENAL NJ 07806-5000
2	COMMANDER US ARMY ARDEC AMSTA AR FSP G M SCHIKSNIS D CARLUCCI PICATINNY ARSENAL NJ 07806-5000
1	COMMANDER US ARMY ARDEC AMSTA AR FSP A P KISATSKY PICATINNY ARSENAL NJ 07806-5000
2	COMMANDER US ARMY ARDEC AMSTA AR CCH C H CHANIN S CHICO PICATINNY ARSENAL NJ 07806-5000
1	COMMANDER US ARMY ARDEC AMSTA AR QAC T D RIGOGLIOSO PICATINNY ARSENAL NJ 07806-5000

NO. OF COPIES	ORGANIZATION
1	COMMANDER US ARMY ARDEC AMSTA AR WET T SACHAR BLDG 172 PICATINNY ARSENAL NJ 07806-5000
9	COMMANDER US ARMY ARDEC AMSTA AR CCH B P DONADIA F DONLON P VALENTI C KNUTSON G EUSTICE S PATEL G WAGNECZ R SAYER F CHANG PICATINNY ARSENAL NJ 07806-5000
6	COMMANDER US ARMY ARDEC AMSTA AR CCL F PUZYCKI R MCHUGH D CONWAY E JAROSZEWSKI R SCHLENNER M CLUNE PICATINNY ARSENAL NJ 07806-5000
5	PM SADARM SFAE GCSS SD COL B ELLIS M DEVINE W DEMASSI J PRITCHARD S HROWNAK PICATINNY ARSENAL NJ 07806-5000
1	US ARMY ARDEC INTELLIGENCE SPECIALIST AMSTA AR WEL F M GUERRIERE PICATINNY ARSENAL NJ 07806-5000

NO. OF COPIES	ORGANIZATION
2	PEO FIELD ARTILLERY SYS SFAE FAS PM H GOLDMAN T MCWILLIAMS PICATINNY ARSENAL NJ 07806-5000
12	PM TMAS SFAE GSSC TMA R MORRIS C KIMKER D GUZIEWICZ E KOPACZ R ROESER R DARCY R KOWALSKI R MCDANOLDS L D ULISSE C ROLLER J MCGREEN B PATTER PICATINNY ARSENAL NJ 07806-5000
1	COMMANDER US ARMY ARDEC AMSTA AR WEA J BRESCIA PICATINNY ARSENAL NJ 07806-5000
1	COMMANDER US ARMY ARDEC PRODUCTION BASE MODERN ACTY AMSMC PBM K PICATINNY ARSENAL NJ 07806-5000
1	COMMANDER US ARMY TACOM PM ABRAMS SFAE ASM AB 6501 ELEVEN MILE RD WARREN MI 48397-5000
1	COMMANDER US ARMY TACOM AMSTA SF WARREN MI 48397-5000

NO. OF COPIES	ORGANIZATION
3	COMMANDER US ARMY TACOM PM TACTICAL VEHICLES SFAE TVL SFAE TVM SFAE TVH 6501 ELEVEN MILE RD WARREN MI 48397-5000
1	COMMANDER US ARMY TACOM PM BFVS SFAE ASM BV 6501 ELEVEN MILE RD WARREN MI 48397-5000
1	COMMANDER US ARMY TACOM PM AFAS SFAE ASM AF 6501 ELEVEN MILE RD WARREN MI 48397-5000
1	COMMANDER US ARMY TACOM PM RDT&E SFAE GCSS W AB J GODELL 6501 ELEVEN MILE RD WARREN MI 48397-5000
2	COMMANDER US ARMY TACOM PM SURV SYS SFAE ASM SS T DEAN SFAE GCSS W GSI M D COCHRAN 6501 ELEVEN MILE RD WARREN MI 48397-5000
1	US ARMY CERL R LAMPO 2902 NEWMARK DR CHAMPAIGN IL 61822

NO. OF COPIES	ORGANIZATION
1	COMMANDER US ARMY TACOM PM SURVIVABLE SYSTEMS SFAE GCSS W GSI H M RYZYI 6501 ELEVEN MILE RD WARREN MI 48397-5000
1	COMMANDER US ARMY TACOM PM BFV SFAE GCSS W BV S DAVIS 6501 ELEVEN MILE RD WARREN MI 48397-5000
1	COMMANDER US ARMY TACOM CHIEF ABRAMS TESTING SFAE GCSS W AB QT T KRASKIEWICZ 6501 ELEVEN MILE RD WARREN MI 48397-5000
1	COMMANDER WATERVLIET ARSENAL SMCWV QAE Q B VANINA BLDG 44 WATERVLIET NY 12189-4050
2	TSM ABRAMS ATZK TS S JABURG W MEINSHAUSEN FT KNOX KY 40121
3	ARMOR SCHOOL ATZK TD R BAUEN J BERG A POMEY FT KNOX KY 40121

NO. OF COPIES	ORGANIZATION
14	COMMANDER US ARMY TACOM AMSTA TR R R MCCLELLAND D THOMAS J BENNETT D HANSEN AMSTA JSK S GOODMAN J FLORENCE K IYER D TEMPLETON A SCHUMACHER AMSTA TR D D OSTBERG L HINOJOSA B RAJU AMSTA CS SF H HUTCHINSON F SCHWARZ WARREN MI 48397-5000
14	BENET LABORATORIES AMSTA AR CCB R FISCELLA M SOJA E KATHE M SCAVULO G SPENCER P WHEELER S KRUPSKI J VASILAKIS G FRIAR R HASENBEIN AMSTA CCB R S SOPOK E HYLAND D CRAYON R DILLON WATERVLIET NY 12189-4050
2	HQ IOC TANK AMMUNITION TEAM AMSIO SMT R CRAWFORD W HARRIS ROCK ISLAND IL 61299-6000

NO. OF COPIES	ORGANIZATION
2	COMMANDER US ARMY AMCOM AVIATION APPLIED TECH DIR J SCHUCK FT EUSTIS VA 23604-5577
1	DIRECTOR US ARMY AMCOM SFAE AV RAM TV D CALDWELL BLDG 5300 REDSTONE ARSENAL AL 35898
2	US ARMY CORPS OF ENGINEERS CERD C T LIU CEW ET T TAN 20 MASS AVE NW WASHINGTON DC 20314
1	US ARMY COLD REGIONS RSCH & ENGRNG LAB P DUTTA 72 LYME RD HANOVER NH 03755
1	SYSTEM MANAGER ABRAMS ATZK TS LTC J H NUNN BLDG 1002 RM 110 FT KNOX KY 40121
1	USA SBCCOM PM SOLDIER SPT AMSSB PM RSS A J CONNORS KANSAS ST NATICK MA 01760-5057
2	USA SBCCOM MATERIAL SCIENCE TEAM AMSSB RSS J HERBERT M SENNETT KANSAS ST NATICK MA 01760-5057

NO. OF COPIES	ORGANIZATION
2	OFC OF NAVAL RESEARCH D SIEGEL CODE 351 J KELLY 800 N QUINCY ST ARLINGTON VA 22217-5660
1	NAVAL SURFACE WARFARE CTR DAHLGREN DIV CODE G06 DAHLGREN VA 22448
1	NAVAL SURFACE WARFARE CTR TECH LIBRARY CODE 323 17320 DAHLGREN RD DAHLGREN VA 22448
1	NAVAL SURFACE WARFARE CTR CRANE DIVISION M JOHNSON CODE 20H4 LOUISVILLE KY 40214-5245
8	DIRECTOR US ARMY NATIONAL GROUND INTELLIGENCE CTR D LEITER M HOLTUS M WOLFE S MINGLEDORF J GASTON W GSTATTENBAUER R WARNER J CRIDER 220 SEVENTH ST NE CHARLOTTESVILLE VA 22091
2	NAVAL SURFACE WARFARE CTR U SORATHIA C WILLIAMS CD 6551 9500 MACARTHUR BLVD WEST BETHESDA MD 20817
2	COMMANDER NAVAL SURFACE WARFARE CTR CARDEROCK DIVISION R PETERSON CODE 2020 M CRITCHFIELD CODE 1730 BETHESDA MD 20084

NO. OF COPIES	ORGANIZATION
8	US ARMY SBCCOM SOLDIER SYSTEMS CENTER BALLISTICS TEAM J WARD W ZUKAS P CUNNIFF J SONG MARINE CORPS TEAM J MACKIEWICZ BUS AREA ADVOCACY TEAM W HASKELL AMSSB RCP SS W NYKVIST S BEAUDOIN KANSAS ST NATICK MA 01760-5019
9	US ARMY RESEARCH OFC A CROWSON J CHANDRA H EVERETT J PRATER R SINGLETON G ANDERSON D STEPP D KISEROW J CHANG PO BOX 12211 RESEARCH TRIANGLE PARK NC 27709-2211
8	NAVAL SURFACE WARFARE CTR J FRANCIS CODE G30 D WILSON CODE G32 R D COOPER CODE G32 J FRAYSSE CODE G33 E ROWE CODE G33 T DURAN CODE G33 L DE SIMONE CODE G33 R HUBBARD CODE G33 DAHLGREN VA 22448
1	NAVAL SEA SYSTEMS CMD D LIESE 2531 JEFFERSON DAVIS HWY ARLINGTON VA 22242-5160
1	NAVAL SURFACE WARFARE CTR M LACY CODE B02 17320 DAHLGREN RD DAHLGREN VA 22448

NO. OF COPIES	ORGANIZATION
2	NAVAL SURFACE WARFARE CTR CARDEROCK DIVISION R CRANE CODE 2802 C WILLIAMS CODE 6553 3A LEGGETT CIR BETHESDA MD 20054-5000
1	EXPEDITIONARY WARFARE DIV N85 F SHOUP 2000 NAVY PENTAGON WASHINGTON DC 20350-2000
1	AFRL MLBC 2941 P ST RM 136 WRIGHT PATTERSON AFB OH 45433-7750
1	AFRL MLSS R THOMSON 2179 12TH ST RM 122 WRIGHT PATTERSON AFB OH 45433-7718
2	AFRL F ABRAMS J BROWN BLDG 653 2977 P ST STE 6 WRIGHT PATTERSON AFB OH 45433-7739
1	WATERWAYS EXPERIMENT D SCOTT 3909 HALLS FERRY RD SC C VICKSBURG MS 39180
5	DIRECTOR LLNL R CHRISTENSEN S DETERESA F MAGNESS M FINGER MS 313 M MURPHY L 282 PO BOX 808 LIVERMORE CA 94550
1	AFRL MLS OL L COULTER 7278 4TH ST BLDG 100 BAY D HILL AFB UT 84056-5205

NO. OF COPIES	ORGANIZATION
1	OSD JOINT CCD TEST FORCE OSD JCCD R WILLIAMS 3909 HALLS FERRY RD VICKSBURG MS 29180-6199
1	DEFENSE NUCLEAR AGENCY INNOVATIVE CONCEPTS DIV 6801 TELEGRAPH RD ALEXANDRIA VA 22310-3398
3	DARPA M VANFOSSEN S WAX L CHRISTODOULOU 3701 N FAIRFAX DR ARLINGTON VA 22203-1714
2	SERDP PROGRAM OFC PM P2 C PELLERIN B SMITH 901 N STUART ST STE 303 ARLINGTON VA 22203
1	FAA MIL HDBK 17 CHAIR L ILCEWICZ 1601 LIND AVE SW ANM 115N RESTON VA 98055
1	US DEPT OF ENERGY OFC OF ENVIRONMENTAL MANAGEMENT P RITZCOVAN 19901 GERMANTOWN RD GERMANTOWN MD 20874-1928
1	DIRECTOR LLNL F ADDESSIO MS B216 PO BOX 1633 LOS ALAMOS NM 87545
1	OAK RIDGE NATIONAL LABORATORY R M DAVIS PO BOX 2008 OAK RIDGE TN 37831-6195

NO. OF COPIES	ORGANIZATION
3	DIRECTOR SANDIA NATIONAL LABS APPLIED MECHANICS DEPT MS 9042 J HANDROCK Y R KAN J LAUFFER PO BOX 969 LIVERMORE CA 94551-0969
1	OAK RIDGE NATIONAL LABORATORY C EBERLE MS 8048 PO BOX 2008 OAK RIDGE TN 37831
1	OAK RIDGE NATIONAL LABORATORY C D WARREN MS 8039 PO BOX 2008 OAK RIDGE TN 37831
5	NIST J DUNKERS M VANLANDINGHAM MS 8621 J CHIN MS 8621 J MARTIN MS 8621 D DUTHINH MS 8611 100 BUREAU DR GAITHERSBURG MD 20899
1	HYDROGEOLOGIC INC SERDP ESTCP SPT OFC S WALSH 1155 HERNDON PKWY STE 900 HERNDON VA 20170
3	NASA LANGLEY RSCH CTR AMSRL VS W ELBER MS 266 F BARTLETT JR MS 266 G FARLEY MS 266 HAMPTON VA 23681-0001
1	NASA LANGLEY RSCH CTR T GATES MS 188E HAMPTON VA 23661-3400
1	FHWA E MUNLEY 6300 GEORGETOWN PIKE MCLEAN VA 22101

NO. OF COPIES	ORGANIZATION
3	CYTEC FIBERITE R DUNNE D KOHLI R MAYHEW 1300 REVOLUTION ST HAVRE DE GRACE MD 21078
1	USDOT FEDERAL RAILRD M FATEH RDV 31 WASHINGTON DC 20590
1	MARINE CORPS INTLLGNC ACTVTY D KOSITZKE 3300 RUSSELL RD STE 250 QUANTICO VA 22134-5011
1	DIRECTOR NATIONAL GRND INTLLGNC CTR IANG TMT 220 SEVENTH ST NE CHARLOTTESVILLE VA 22902-5396
1	SIOUX MFG B KRIEL PO BOX 400 FT TOTTEN ND 58335
2	3TEX CORPORATION A BOGDANOVICH J SINGLETARY 109 MACKENAN DR CARY NC 27511
1	3M CORPORATION J SKILDUM 3M CENTER BLDG 60 IN 01 ST PAUL MN 55144-1000
1	DIRECTOR DEFENSE INTLLGNC AGNCY TA 5 K CRELLING WASHINGTON DC 20310
1	ADVANCED GLASS FIBER YARNS T COLLINS 281 SPRING RUN LANE STE A DOWNINGTON PA 19335

NO. OF COPIES	ORGANIZATION
1	COMPOSITE MATERIALS INC D SHORTT 19105 63 AVE NE PO BOX 25 ARLINGTON WA 98223
1	JPS GLASS L CARTER PO BOX 260 SLATER RD SLATER SC 29683
1	COMPOSITE MATERIALS INC R HOLLAND 11 JEWEL CT ORINDA CA 94563
1	COMPOSITE MATERIALS INC C RILEY 14530 S ANSON AVE SANTA FE SPRINGS CA 90670
2	SIMULA J COLTMAN R HUYETT 10016 S 51ST ST PHOENIX AZ 85044
2	PROTECTION MATERIALS INC M MILLER F CRILLEY 14000 NW 58 CT MIAMI LAKES FL 33014
2	FOSTER MILLER M ROYLANCE W ZUKAS 195 BEAR HILL RD WALTHAM MA 02354-1196
1	ROM DEVELOPMENT CORP R O MEARA 136 SWINEBURNE ROW BRICK MARKET PLACE NEWPORT RI 02840

NO. OF COPIES	ORGANIZATION
2	TEXTRON SYSTEMS T FOLTZ M TREASURE 1449 MIDDLESEX ST LOWELL MA 01851
1	O GARA HESS & EISENHARDT M GILLESPIE 9113 LESAINT DR FAIRFIELD OH 45014
2	MILLIKEN RSCH CORP H KUHN M MACLEOD PO BOX 1926 SPARTANBURG SC 29303
1	CONNEAUGHT INDUSTRIES INC J SANTOS PO BOX 1425 COVENTRY RI 02816
1	BATTELLE NATICK OPNS B HALPIN 209 W CENTRAL ST STE 302 NATICK MA 01760
1	ARMTEC DEFENSE PRODUCTS S DYER 85 901 AVE 53 PO BOX 848 COACHELLA CA 92236
1	NATIONAL COMPOSITE CENTER T CORDELL 2000 COMPOSITE DR KETTERING OH 45420
3	PACIFIC NORTHWEST LAB M SMITH G VAN ARSDALE R SHIPPELL PO BOX 999 RICHLAND WA 99352
2	AMOCO PERFORMANCE PRODUCTS M MICHNO JR J BANISAUKAS 4500 MCGINNIS FERRY RD ALPHARETTA GA 30202-3944

NO. OF COPIES	ORGANIZATION
8	ALLIANT TECHSYSTEMS INC C CANDLAND MN11 2830 C AAKHUS MN11 2830 B SEE MN11 2439 N VLAHAKUS MN11 2145 R DOHRN MN11 2830 S HAGLUND MN11 2439 M HISSONG MN11 2830 D KAMDAR MN11 2830 600 SECOND ST NE HOPKINS MN 55343-8367
1	SAIC M PALMER 1410 SPRING HILL RD STE 400 MS SH4 5 MCLEAN VA 22102
1	SAIC G CHRYSSOMALLIS 3800 W 80TH ST STE 1090 BLOOMINGTON MN 55431
1	AAI CORPORATION T G STASTNY PO BOX 126 HUNT VALLEY MD 21030-0126
1	APPLIED COMPOSITES W GRISCH 333 NORTH SIXTH ST ST CHARLES IL 60174
1	CUSTOM ANALYTICAL ENG SYS INC A ALEXANDER 13000 TENSOR LANE NE FLINTSTONE MD 21530
3	ALLIANT TECHSYSTEMS INC J CONDON E LYNAM J GERHARD WV01 16 STATE RT 956 PO BOX 210 ROCKET CENTER WV 26726-0210
1	OFC DEPUTY UNDER SEC DEFNS J THOMPSON 1745 JEFFERSON DAVIS HWY CRYSTAL SQ 4 STE 501 ARLINGTON VA 22202

NO. OF COPIES	ORGANIZATION
1	PROJECTILE TECHNOLOGY INC 515 GILES ST HAVRE DE GRACE MD 21078
5	AEROJET GEN CORP D PILLASCH T COULTER C FLYNN D RUBAREZUL M GREINER 1100 WEST HOLLYVALE ST AZUSA CA 91702-0296
3	HEXCEL INC R BOE PO BOX 18748 SALT LAKE CITY UT 84118
1	HERCULES INC HERCULES PLAZA WILMINGTON DE 19894
1	BRIGS COMPANY J BACKOFEN 2668 PETERBOROUGH ST HERNDON VA 22071-2443
1	ZERNOW TECHNICAL SERVICES L ZERNOW 425 W BONITA AVE STE 208 SAN DIMAS CA 91773
1	GENERAL DYNAMICS OTS L WHITMORE 10101 NINTH ST NORTH ST PETERSBURG FL 33702
3	GENERAL DYNAMICS OTS FLINCHBAUGH DIV E STEINER B STEWART T LYNCH PO BOX 127 RED LION PA 17356
1	GKN AEROSPACE D OLDS 15 STERLING DR WALLINGFORD CT 06492

NO. OF COPIES	ORGANIZATION
5	SIKORSKY AIRCRAFT G JACARUSO T CARSTENSAN B KAY S GARBO MS S330A J ADELMANN 6900 MAIN ST PO BOX 9729 STRATFORD CT 06497-9729
1	PRATT & WHITNEY C WATSON 400 MAIN ST MS 114 37 EAST HARTFORD CT 06108
1	AEROSPACE CORP G HAWKINS M4 945 2350 E EL SEGUNDO BLVD EL SEGUNDO CA 90245
2	CYTEC FIBERITE M LIN W WEB 1440 N KRAEMER BLVD ANAHEIM CA 92806
1	UDLP G THOMAS PO BOX 58123 SANTA CLARA CA 95052
2	UDLP R BARRETT MAIL DROP M53 V HORVATICH MAIL DROP M53 328 W BROKAW RD SANTA CLARA CA 95052-0359
3	UDLP GROUND SYSTEMS DIVISION M PEDRAZZI MAIL DROP N09 A LEE MAIL DROP N11 M MACLEAN MAIL DROP N06 1205 COLEMAN AVE SANTA CLARA CA 95052
4	UDLP R BRYNSVOLD P JANKE MS 170 4800 EAST RIVER RD MINNEAPOLIS MN 55421-1498

NO. OF COPIES	ORGANIZATION
1	UDLP D MARTIN PO BOX 359 SANTA CLARA CA 95052
2	BOEING DFNSE & SPACE GP W HAMMOND S 4X55 J RUSSELL S 4X55 PO BOX 3707 SEATTLE WA 98124-2207
2	BOEING ROTORCRAFT P MINGURT P HANDEL 800 B PUTNAM BLVD WALLINGFORD PA 19086
1	BOEING DOUGLAS PRODUCTS DIV L J HART SMITH 3855 LAKEWOOD BLVD D800 0019 LONG BEACH CA 90846-0001
1	LOCKHEED MARTIN SKUNK WORKS D FORTNEY 1011 LOCKHEED WAY PALMDALE CA 93599-2502
1	LOCKHEED MARTIN R FIELDS 1195 IRWIN CT WINTER SPRINGS FL 32708
1	MATERIALS SCIENCES CORP G FLANAGAN 500 OFC CENTER DR STE 250 FT WASHINGTON PA 19034
1	NORTHRUP GRUMMAN CORP ELECTRONIC SENSORS & SYSTEMS DIV E SCHOCH MS V 16 1745A W NURSERY RD LINTHICUM MD 21090
1	GDLS DIVISION D BARTLE PO BOX 1901 WARREN MI 48090

NO. OF COPIES	ORGANIZATION
2	GDLS D REES M PASIK PO BOX 2074 WARREN MI 48090-2074
1	GDLS MUSKEGON OPERATIONS W SOMMERS JR 76 GETTY ST MUSKEGON MI 49442
1	GENERAL DYNAMICS AMPHIBIOUS SYS SURVIVABILITY LEAD G WALKER 991 ANNAPOLIS WAY WOODBRIDGE VA 22191
6	INST FOR ADVANCED TECH H FAIR I MCNAB P SULLIVAN S BLESS W REINECKE C PERSAD 3925 W BRAKER LN STE 400 AUSTIN TX 78759-5316
2	CIVIL ENGR RSCH FOUNDATION PRESIDENT H BERNSTEIN R BELLE 1015 15TH ST NW STE 600 WASHINGTON DC 20005
1	ARROW TECH ASSO 1233 SHELBURNE RD STE D8 SOUTH BURLINGTON VT 05403-7700
1	R EICHELBERGER CONSULTANT 409 W CATHERINE ST BEL AIR MD 21014-3613

NO. OF COPIES	ORGANIZATION
1	UCLA MANE DEPT ENGR IV H T HAHN LOS ANGELES CA 90024-1597
2	UNIV OF DAYTON RESEARCH INST R Y KIM A K ROY 300 COLLEGE PARK AVE DAYTON OH 45469-0168
1	UMASS LOWELL PLASTICS DEPT N SCHOTT 1 UNIVERSITY AVE LOWELL MA 01854
1	IIT RESEARCH CENTER D ROSE 201 MILL ST ROME NY 13440-6916
1	GA TECH RSCH INST GA INST OF TCHNLGY P FRIEDERICH ATLANTA GA 30392
1	MICHIGAN ST UNIV MSM DEPT R AVERILL 3515 EB EAST LANSING MI 48824-1226
1	UNIV OF WYOMING D ADAMS PO BOX 3295 LARAMIE WY 82071
2	PENN STATE UNIV R MCNITT C BAKIS 212 EARTH ENGR SCIENCES BLDG UNIVERSITY PARK PA 16802
1	PENN STATE UNIV R S ENGEL 245 HAMMOND BLDG UNIVERSITY PARK PA 16801

NO. OF COPIES	ORGANIZATION
1	PURDUE UNIV SCHOOL OF AERO & ASTRO C T SUN W LAFAYETTE IN 47907-1282
1	STANFORD UNIV DEPT OF AERONAUTICS & AEROBALLISTICS S TSAI DURANT BLDG STANFORD CA 94305
1	UNIV OF MAIN ADV STR & COMP LAB R LOPEZ ANIDO 5793 AEWC BLDG ORONO ME 04469-5793
1	JOHNS HOPKINS UNIV APPLIED PHYSICS LAB P WIENHOLD 11100 JOHNS HOPKINS RD LAUREL MD 20723-6099
1	UNIV OF DAYTON J M WHITNEY COLLEGE PARK AVE DAYTON OH 45469-0240
5	UNIV OF DELAWARE CTR FOR COMPOSITE MTRLS J GILLESPIE M SANTARE S YARLAGADDA S ADVANI D HEIDER 201 SPENCER LABORATORY NEWARK DE 19716
1	DEPT OF MATERIALS SCIENCE & ENGINEERING UNIVERSITY OF ILLINOIS AT URBANA CHAMPAIGN J ECONOMY 1304 WEST GREEN ST 115B URBANA IL 61801

NO. OF COPIES	ORGANIZATION
1	NORTH CAROLINA STATE UNIV CIVIL ENGINEERING DEPT W RASDORF PO BOX 7908 RALEIGH NC 27696-7908
1	UNIV OF MARYLAND DEPT OF AEROSPACE ENGNRNG A J VIZZINI COLLEGE PARK MD 20742
3	UNIV OF TEXAS AT AUSTIN CTR FOR ELECTROMECHANICS J PRICE A WALLS J KITZMILLER 10100 BURNET RD AUSTIN TX 78758-4497
3	VA POLYTECHNICAL INST & STATE UNIV DEPT OF ESM M W HYER K REIFSNIDER R JONES BLACKSBURG VA 24061-0219
1	DREXEL UNIV A S D WANG 32ND & CHESTNUT ST PHILADELPHIA PA 19104
1	SOUTHWEST RSCH INST ENGR & MATL SCIENCES DIV J RIEGEL 6220 CULEBRA RD PO DRAWER 28510 SAN ANTONIO TX 78228-0510

ABERDEEN PROVING GROUND

NO. OF COPIES	ORGANIZATION
1	US ARMY MATERIEL SYSTEMS ANALYSIS ACTIVITY P DIETZ 392 HOPKINS RD AMXSY TD APG MD 21005-5071
1	DIRECTOR US ARMY RESEARCH LAB AMSRL OP AP L APG MD 21005-5066

ABERDEEN PROVING GROUND (CONT)

NO. OF COPIES	ORGANIZATION
91	DIR USARL AMSRL CI AMSRL CI H W STUREK AMSRL CI S A MARK AMSRL CS IO FI M ADAMSON AMSRL SL BA AMSRL SL BL D BELY R HENRY AMSRL SL BG AMSRL SL I AMSRL WM J SMITH AMSRL WM B A HORST AMSRL WM BA D LYON AMSRL WM BC P PLOSTINS J NEWILL S WILKERSON A ZIELINSKI AMSRL WM BD B FORCH R FIFER R PESCE RODRIGUEZ B RICE AMSRL WM BE C LEVERITT AMSRL WM BF J LACETERA AMSRL WM BR C SHOEMAKER J BORNSTEIN AMSRL WM M D VIECHNICKI G HAGNAUER J MCCAULEY AMSRL WM MA L GHIORSE S MCKNIGHT AMSRL WM MB B FINK J BENDER T BOGETTI R BOSSOLI L BURTON

NO. OF COPIES	ORGANIZATION
	ABERDEEN PROVING GROUND (CONT)
	K BOYD S CORNELISON P DEHMER R DOOLEY W DRYSDALE G GAZONAS S GHIORSE D GRANVILLE D HOPKINS C HOPPEL D HENRY R KASTE M KLUSEWITZ M LEADORE R LIEB E RIGAS J SANDS D SPAGNUOLO W SPURGEON J TZENG E WETZEL A FRYDMAN AMRSL WM MC J BEATTY E CHIN J MONTGOMERY A WERECZCAK J LASALVIA J WELLS AMSRL WM MD W ROY S WALSH AMSRL WM T B BURNS M ZOLTOSKI AMSRL WM TA W GILLICH T HAVEL J RUNYEON M BURKINS E HORWATH B GOOCH W BRUCHEY M NORMANDIA

NO. OF COPIES	ORGANIZATION
	ABERDEEN PROVING GROUND (CONT)
	AMRSL WM TB D KOOKER P BAKER AMSRL WM TC R COATES AMSRL WM TD A DAS GUPTA T HADUCH T MOYNIHAN F GREGORY M RAFTENBERG M BOTELER T WEERASOORIYA D DANDEKAR A DIETRICH AMSRL WM TE A NIILER J POWELL AMSRL SS SD H WALLACE AMSRL SS SE DS R REYZER R ATKINSON

NO. OF COPIES	ORGANIZATION
1	LTD R MARTIN MERL TAMWORTH RD HERTFORD SG13 7DG UK
1	SMC SCOTLAND P W LAY DERA ROSYTH ROSYTH ROYAL DOCKYARD DUNFERMLINE FIFE KY 11 2XR UK
1	CIVIL AVIATION ADMINSTRATION T GOTTESMAN PO BOX 8 BEN GURION INTERNL AIRPORT LOD 70150 ISRAEL
1	AEROSPATIALE S ANDRE A BTE CC RTE MD132 316 ROUTE DE BAYONNE TOULOUSE 31060 FRANCE
1	DRA FORT HALSTEAD P N JONES SEVEN OAKS KENT TN 147BP UK
1	DEFENSE RESEARCH ESTAB VALCARTIER F LESAGE COURCELETTE QUEBEC COA IRO CANADA
1	SWISS FEDERAL ARMAMENTS WKS W LANZ ALLMENDSTRASSE 86 3602 THUN SWITZERLAND
1	DYNAMEC RESEARCH AB AKE PERSSON BOX 201 SE 151 23 SODERTALJE SWEDEN

NO. OF COPIES	ORGANIZATION
1	ISRAEL INST OF TECHNOLOGY S BODNER FACULTY OF MECHANICAL ENGR HAIFA 3200 ISRAEL
1	DSTO MATERIALS RESEARCH LAB NAVAL PLATFORM VULNERABILITY SHIP STRUCTURES & MTRLS DIV N BURMAN PO BOX 50 ASCOT VALE VICTORIA AUSTRALIA 3032
1	ECOLE ROYAL MILITAIRE E CELENS AVE DE LA RENAISSANCE 30 1040 BRUXELLE BELGIQUE
1	DEF RES ESTABLISHMENT VALCARTIER A DUPUIS 2459 BOULEVARD PIE XI NORTH VALCARTIER QUEBEC CANADA PO BOX 8800 COURCELETTE GOA IRO QUEBEC CANADA
1	INSTITUT FRANCO ALLEMAND DE RECHERCHES DE SAINT LOUIS DE M GIRAUD 5 RUE DU GENERAL CASSAGNOU BOITE POSTALE 34 F 68301 SAINT LOUIS CEDEX FRANCE
1	ECOLE POLYTECH J MANSON DMX LTC CH 1015 LAUSANNE SWITZERLAND

NO. OF COPIES	ORGANIZATION
1	TNO PRINS MAURITS LABORATORY R IJSSELSTEIN LANGE KLEIWEG 137 PO BOX 45 2280 AA RIJSWIJK THE NETHERLANDS
2	FOA NATL DEFENSE RESEARCH ESTAB DIR DEPT OF WEAPONS & PROTECTION B JANZON R HOLMLIN S 172 90 STOCKHOLM SWEDEN
2	DEFENSE TECH & PROC AGENCY GROUND I CREWTHER GENERAL HERZOG HAUS 3602 THUN SWITZERLAND
1	MINISTRY OF DEFENCE RAFAEL ARMAMENT DEVELOPMENT AUTH M MAYSELESS PO BOX 2250 HAIFA 31021 ISRAEL
1	TNO DEFENSE RESEARCH I H PASMAN POSTBUS 6006 2600 JA DELFT THE NETHERLANDS
1	B HIRSCH TACHKEMONY ST 6 NETAMUA 42611 ISRAEL
1	DEUTSCHE AEROSPACE AG DYNAMICS SYSTEMS M HELD PO BOX 1340 D 86523 SCHROBENHAUSEN GERMANY

NO. OF COPIES	ORGANIZATION
10	E SAETHER 104 MICHAEL PLACE YORKTOWN VA 23692

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. AGENCY USE ONLY *(Leave blank)*	2. REPORT DATE	3. REPORT TYPE AND DATES COVERED
	July 2001	Final, November 1995 - November 1997

4. TITLE AND SUBTITLE
RESTRAN: Residual Strength Analysis of Impact Damaged Composite Laminates
Volume II: User's Manual

5. FUNDING NUMBERS
622105AH84

6. AUTHOR(S)
Erik Saether

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
U.S. Army Research Laboratory
ATTN: AMSRL-WM-MB
Aberdeen Proving Ground, MD 21005-5069

8. PERFORMING ORGANIZATION REPORT NUMBER
ARL-TR-2550

9. SPONSORING/MONITORING AGENCY NAMES(S) AND ADDRESS(ES)

10.SPONSORING/MONITORING AGENCY REPORT NUMBER

11. SUPPLEMENTARY NOTES

12a. DISTRIBUTION/AVAILABILITY STATEMENT
Approved for public release; distribution is unlimited.

12b. DISTRIBUTION CODE

13. ABSTRACT ***(Maximum 200 words)***

A general predictive methodology for determining residual strength in impact damaged composite laminates has been developed and incorporated into a computer code designated RESTRAN (REsidual STRength ANalysis). RESTRAN is a finite element based design tool that can analyze composite structures with arbitrary three-dimensional (3-D) geometry, loading and support conditions, material properties, and initial material and delamination damage. Material failure modes are predicted using a robust suite of failure criteria and damage laws. Structural failure due to sequential sublaminate buckling of delaminated layers is also accounted for. A progressive failure analysis is performed until ultimate structural failure is predicted, thereby yielding an estimate of the residual strength. This report contains a user's manual for the RESTRAN program with complete descriptions of input statements and program output. Several examples are shown to illustrate the use of the RESTRAN computer code.

14. SUBJECT TERMS
residual strength, multiple delaminations, material failure, impact damage

15. NUMBER OF PAGES
140

16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
UNCLASSIFIED	UNCLASSIFIED	UNCLASSIFIED	UL

NSN 7540-01-280-5500

Standard Form 298 (Rev. 2-89)
Prescribed by ANSI Std. 239-18 298-102

INTENTIONALLY LEFT BLANK.

USER EVALUATION SHEET/CHANGE OF ADDRESS

This Laboratory undertakes a continuing effort to improve the quality of the reports it publishes. Your comments/answers to the items/questions below will aid us in our efforts.

1. ARL Report Number/Author ARL-TR-2550 (POC: Fink) ______ Date of Report July 2001 ______

2. Date Report Received ______

3. Does this report satisfy a need? (Comment on purpose, related project, or other area of interest for which the report will be used.) ______

______

4. Specifically, how is the report being used? (Information source, design data, procedure, source of ideas, etc.) ______

______

______

5. Has the information in this report led to any quantitative savings as far as man-hours or dollars saved, operating costs avoided, or efficiencies achieved, etc? If so, please elaborate. ______

______

______

6. General Comments. What do you think should be changed to improve future reports? (Indicate changes to organization, technical content, format, etc.) ______

______

______

______

CURRENT ADDRESS

______
Organization

______
Name E-mail Name

______
Street or P.O. Box No.

______
City, State, Zip Code

7. If indicating a Change of Address or Address Correction, please provide the Current or Correct address above and the Old or Incorrect address below.

OLD ADDRESS

______
Organization

______
Name

______
Street or P.O. Box No.

______
City, State, Zip Code

(Remove this sheet, fold as indicated, tape closed, and mail.)

(DO NOT STAPLE)